

® 2024 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Command Reference

Ansys

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2024 R1
January 2024

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2024 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. About This Reference 1
1.1. Conventions Used in This Reference 1
1.1.1. Applicable Products 2
1.1.2. Valid Products 2
1.2. Terminology 3
1.3. Command Characteristics 5
1.3.1. Data Input 5
1.3.2. Free-Format Input 5
1.3.3. Nonrestrictive Data Input 5
1.3.4. Condensed Data Input 6
1.3.5. Units 6
1.3.6. Command and Argument Defaults 7
1.3.7. File Names 7
1.3.8. Star and Slash Commands 8
2. Command Groupings 11
2.1. SESSION Commands 11
2.2. DATABASE Commands 13
2.3. GRAPHICS Commands 16
2.4. APDL Commands 19
2.5. PREP7 Commands 22
2.6. SOLUTION Commands 38
2.7. POST1 Commands 51
2.8. POST26 Commands 58
2.9. AUX2 Commands 61
2.10. AUX3 Commands 61
2.11. AUX12 Commands 62
2.12. AUX15 Commands 63
2.13. Mapping Processor Commands 63
3. Command Dictionary 65
I. Connection Commands 69
II. Plugin Commands 79
III. A Commands 81
IV. B Commands 217
V. C Commands 283
VI. D Commands 483
VII. E Commands 591
VIII. F Commands 729
IX. G Commands 787
X. H Commands 885
XI. I Commands 921
XII. J Commands 965
XIII. K Commands 969
XIV. L Commands 1005
XV. M Commands 1097
XVI. N Commands 1207
XVII. O Commands 1307
XVIII. P Commands 1347
XIX. Q Commands 1559
XX. R Commands 1565

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

XXI. S Commands 1643
XXII. T Commands 1897
XXIII. U Commands 2037
XXIV. V Commands 2069
XXV. W Commands 2177
XXVI. X Commands 2191
XXVII. Y Commands 2201
XXVIII. Z Commands 2203

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

1. RCL Lumped Circuit 253
2. Example Command Actions 599
3. Boundary Condition Key Behavior 600
4. Input Signal 748
5. Amplitude Spectrum of the FFT 749
6. Signal Obtained with the Filter 749
7. Basic Ocean Data Type Components 1309
8. Ocean Zone Types (Specified via *ZoneType*) 1323
9. Usage Example: Positive and Negative *NSECT* Values 1635
10. Usage Example: Extrusions with *Axis* = ZEXT and CEXT 1636
11. Independent and Dependent Facets in a Model with Planar Symmetry Employing View Factor Condensation 1636
12. Independent and Dependent Facets in a Model with Cyclic Symmetry Employing View Factor Condensation 1637
13. Independent and Dependent Facets for a Model Built by Extrusions Employing View Factor Condensation 1637
14. Usage Example: *Option* = COND 1638
15. Offsets for a CHAN Section Subtype 1686
16. Two-hole Box Section 1695
17. Cell Mesh for the Two-hole Box Section 1695
18. Coordinate System for Load Application on the Faces of 3D Solid and Shell Elements 1751
19. Load Direction in the Default Coordinate System 1753
20. Default Coordinate System of Surface Load on the Edge of a 3D Shell Element 1754
21. Projected Coordinate System of Surface Load on the Edge of a 3D Shell Element 1754
22. Positive Tangential Load on the Edge of a Plane Element 1755
23. Follower Load Behavior in the Projected Coordinate System 1756
24. Follower Load Behavior for a User-Defined Orientation 1756
25. Follower Load Behavior on the Edges of 3D Shell and 2D Elements 1757
26. Virtual Shell Elements Following the Contact Interface Edge 1793
27. Virtual Shell Elements Overlapping Target Elements 1793
28. View Factor Update Timing 2112
29. Examples of Radiation Problems that Undergo an Enclosure Status Change 2114
30. View Factor Updates and Reciprocity 2115

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

2.1. Run Controls 11
2.2. Processor Entry 12
2.3. Files 12
2.4. List Controls 13
2.5. Setup 13
2.6. Selecting 14
2.7. Components 14
2.8. Working Plane 15
2.9. Coordinate System 15
2.10. Picking 15
2.11. Setup 16
2.12. Views 17
2.13. Scaling 17
2.14. Style 17
2.15. Labeling 18
2.16. Graphs 18
2.17. Annotation 18
2.18. Parameter Definition 19
2.19. Macro Files 19
2.20. Encryption/Decryption 20
2.21. Abbreviations 20
2.22. Array Parameters 20
2.23. Process Controls 21
2.24. Matrix Operations (APDL Math) 21
2.25. Database 22
2.26. Element Type 23
2.27. Real Constants 23
2.28. Materials 24
2.29. Material Data Tables 24
2.30. Primitives 25
2.31. Keypoints 25
2.32. Hard Points 26
2.33. Lines 26
2.34. Areas 27
2.35. Volumes 28
2.36. Booleans 28
2.37. Meshing 29
2.38. Nodes 31
2.39. Elements 32
2.40. Superelements 34
2.41. Coupled Degrees of Freedom 34
2.42. Constraint Equations 34
2.43. Status 35
2.44. Sections 35
2.45. Morphing 37
2.46. Artificially Matched Layers 37
2.47. Special Purpose 37
2.48. Analysis Options 38
2.49. Nonlinear Options 40

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.50. Dynamic Options 40
2.51. Spectrum Options 41
2.52. Load Step Options 43
2.53. Solid Constraints 43
2.54. Solid Forces 44
2.55. Solid Surface Loads 44
2.56. Solid Body Loads 44
2.57. Inertia 45
2.58. Miscellaneous Loads 45
2.59. Load Step Operations 46
2.60. Master Degrees of Freedom 46
2.61. Rezoning 46
2.62. 2D to 3D Analysis 47
2.63. Birth and Death 47
2.64. FE Constraints 47
2.65. FE Forces 48
2.66. FE Surface Loads 48
2.67. FE Body Loads 48
2.68. Ocean 49
2.69. Status 49
2.70. Radiosity 50
2.71. Additive Manufacturing 50
2.72. Setup 51
2.73. Controls 51
2.74. Results 51
2.75. Element Table 52
2.76. Listing 53
2.77. Animation 53
2.78. Path Operations 54
2.79. Surface Operations 54
2.80. Load Case Calculations 55
2.81. Magnetics Calculations 55
2.82. Trace Points 56
2.83. Special Purpose 56
2.84. Status 58
2.85. Failure Criteria 58
2.86. Setup 58
2.87. Controls 59
2.88. Operations 59
2.89. Display 60
2.90. Listing 60
2.91. Special Purpose 60
2.92. Status 60
2.93. Binary File Dump 61
2.94. Binary File Manipulation 61
2.95. Results Files 61
2.96. General Radiation 62
2.97. Radiation Matrix Method 62
2.98. Radiosity Solver 62
2.99. IGES 63
2.100. Pressure Mapping 63

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

101. ANSOL - General Result Item and Component Labels 151
102. ANSOL - Selected Result (SRES) Component Labels 153
103. ASEL - Valid Item and Component Labels 195
104. Contact Status Adjusted in Linear Perturbation Analysis via CNKMOD 393
105. DESOL - Valid Item and Component Labels 521
106. *DIM - Primary Variables 533
107. DNSOL - Valid Item and Component Labels 565
108. Examples: End-Release Conditions at a Junction Node when *KJCT* = 1 and *KCHECK* = 0 639
109. Examples: Using *KCHECK* with *TOLERANCE* = 20 640
110. ESEL - Valid Item and Component Labels 663
111. ESOL - General Result Item and Component Labels 673
112. ESOL - Selected Result (SRES) Component Labels 679
113. ETABLE - General Result Item and Component Labels 690
114. ETABLE - Selected Result (SRES) Component Labels 698
115. *GET General Items, Entity = ACTIVE 799
116. *GET General Items, Entity = CMD 800
117. *GET General Items, Entity = COMP 800
118. *GET General Items, Entity = GRAPH 801
119. *GET General Items, Entity = PARM 802
120. *GET General Items, Entity = ***TBTYPE*** 803
121. *GET Preprocessing Items, Entity = ACTIVE 804
122. *GET Preprocessing items, Entity = AREA 805
123. *GET Preprocessing Items, Entity = AXIS 806
124. *GET Preprocessing Items, Entity = CDSY 806
125. *GET Preprocessing Items, Entity = CE 807
126. *GET Preprocessing Items, Entity = CMPB 807
127. *GET Preprocessing Items, Entity = CP 808
128. *GET Preprocessing Items, Entity = CSEC 808
129. *GET Preprocessing Items, Entity = ELEM 809
130. *GET Preprocessing Items, Entity = ETYP 811
131. *GET Preprocessing Items, Entity = GCN 811
132. *GET Preprocessing Items, Entity = GENB 812
133. *GET Preprocessing Items, Entity = GENS 814
134. *GET Preprocessing Items, Entity = KP 815
135. *GET Preprocessing Items, Entity = LINE 816
136. *GET Preprocessing Items, Entity = LINK 817
137. *GET Preprocessing Items, Entity = MAT 818
138. *GET Preprocessing Items, Entity = MPLAB 818
139. *GET Preprocessing Items, Entity = NODE 818
140. *GET Preprocessing Items, Entity = OCEAN 820
141. *GET Preprocessing Items, Entity = OCZONE 821
142. *GET Preprocessing Items, Entity = PIPE 821
143. *GET Preprocessing Items, Entity = PART 822
144. *GET Preprocessing Items, Entity = RCON 822
145. *GET Preprocessing Items, Entity = REIN 823
146. *GET Preprocessing Items, Entity = SCTN 823
147. *GET Preprocessing Items, Entity = SECP 824
148. *GET Preprocessing Items, Entity = SHEL 825
149. *GET Preprocessing Items, Entity = TBFT 826
150. *GET Preprocessing Items, Entity = TBLAB 827
151. *GET Preprocessing Items, Entity = VOLU 828

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

152. *GET Solution Items, Entity = ACTIVE 829
153. *GET Solution Items, Entity = ELEM 830
154. *GET Solution Items, Entity = MODE 831
155. *GET Solution Items, Entity = DDAM 832
156. *GET Postprocessing Items, Entity = ACTIVE 833
157. *GET Postprocessing Items, Entity = ACUS 834
158. *GET Postprocessing Items, Entity = CAMP 835
159. *GET Postprocessing Items, Entity = CINT 836
160. *GET Postprocessing Items, Entity = CYCCALC 838
161. *GET Postprocessing Items, Entity = ELEM 838
162. *GET Postprocessing Items, Entity = ETAB 839
163. *GET Postprocessing Items, Entity = FSUM 840
164. *GET Postprocessing Items, Entity = GSRESULT 840
165. *GET Postprocessing Items, Entity = MEMBER 840
166. *GET Postprocessing Items, Entity = NODE 840
167. *GET Postprocessing Items, Entity = PATH 848
168. *GET Postprocessing Items, Entity = PLNSOL 849
169. *GET Postprocessing Items, Entity = PRENERGY 849
170. *GET Postprocessing Items, Entity = PRERR 850
171. *GET Postprocessing Items, Entity = RAD 850
172. *GET Postprocessing Items, Entity = RSTMAC 851
173. *GET Postprocessing Items, Entity = SECR 852
174. *GET Postprocessing Items, Entity = SECTION 855
175. *GET Postprocessing Items, Entity = SORT 856
176. *GET Postprocessing Items, Entity = SSUM 856
177. *GET Postprocessing Items, Entity = VARI 856
178. *GET Postprocessing Items, Entity = XFEM 857
179. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = AFT 892
180. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = NR 892
181. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = TRTOL 893
182. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = CONTSET 893
183. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = CONTTERM 895
184. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = UINIT 895
185. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = LINEAR 897
186. Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = SCAL 898
187. Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = MSUP 913
188. Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = VT or VTPA 915
189. Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = KRYLOV 915
190. Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = HBM 916
191. JSOL - Valid Item and Component Labels 967
192. KSEL - Valid Item and Component Labels 997
193. LSEL - Valid Item and Component Labels 1076
194. NLHIST - Valid NSOL Item and Component Labels 1235
195. NLHIST - Valid ESOL Item and Component Labels 1235
196. NLHIST - Valid Contact (PAIR or GCN) Item and Component Labels 1237
197. NLHIST - Valid Result Section (RSEC) Item and Component Labels 1239
198. NSEL - Valid Item and Component Labels 1268
199. NSEL - Valid Item and Component Labels for Nodal DOF Result Values 1270
200. NSEL - Valid Item and Component Labels for Element Result Values 1270
201. Supported Elements 1278
202. Supported Total Mass Distribution Methods 1279

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

203. Supported Units for Direct Input of Mass Density 1279
204. NSOL - Valid Item and Component Labels 1281
205. NSORT - Valid Item and Component Labels 1284
206. NSORT - Valid Item and Component Labels for Nodal DOF Result Values 1284
207. NSORT - Valid Item and Component Labels for Element Result Values 1284
208. OSRESULT - Item and Component Labels 1328
209. PDEF - Valid Item and Component Labels 1373
210. Adjusted Contact Status when PERTURB Command Is Issued 1383
211. Adjusted Contact Status when Both CNKMOD and PERTURB Are Issued 1383
212. PLESOL - General Result Item and Component Labels 1402
213. PLESOL - Selected Result (SRES) Component Labels 1411
214. PLNSOL - Valid Item and Component Labels 1427
215. PLNSOL - Selected Result (SRES) Component Labels 1434
216. PLSECT - Valid Item and Component Labels 1441
217. PLTRAC - Valid Item and Component Labels 1444
218. PLVECT - Valid Item Labels 1447
219. PRESOL - General Result Item and Component Labels 1482
220. PRESOL - Selected Result (SRES) Component Labels 1489
221. PRNSOL - General Result Item and Component Labels 1507
222. PRNSOL - Selected Result (SRES) Component Labels 1514
223. PRVECT - Valid Item and Component Labels 1524
224. /PSF - Valid Item and Component Labels 1539
225. RFORCE - Valid Item and Component Labels 1607
226. SMRTSIZE - Argument Values for h-elements 1813
227. Set Operations 1971
228. Set Operations 1974
229. Set Operations 1978
230. Set Operations 1982
231. Creep Options 1983
232. Set Operations 1986
233. *VGET PREP7 Items, Entity = NODE 2120
234. *VGET PREP7 Items, Entity = ELEM 2120
235. *VGET PREP7 Items, Entity = KP 2121
236. *VGET PREP7 Items, Entity = LINE 2121
237. *VGET PREP7 Items, Entity = AREA 2121
238. *VGET PREP7 Items, Entity = VOLU 2122
239. *VGET PREP7 Items, Entity = CDSY 2122
240. *VGET PREP7 Items, Entity = RCON 2123
241. *VGET PREP7 Items, Entity = TLAB 2123
242. *VGET POST1 Items, Entity = CYCCALC 2124
243. *VGET POST1 Items, Entity = ELEM 2125
244. *VGET POST1 Items, Entity = MEMBER 2125
245. *VGET POST1 Items, Entity = NODE, Element Nodal Results 2125
246. *VGET POST1 Items, Entity = NODE, Nodal Degree of Freedom Results 2129
247. *VPUT - POST1 Items 2150
248. VSEL - Valid Item and Component Labels 2165

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: About This Reference

Welcome to the *Command Reference* (p. 1). This reference contains a complete dictionary of detailed Mechanical APDL command descriptions, arranged in alphabetical order. It is the definitive resource for correct command usage, providing comprehensive specifications for every argument of every command.

The *Command Reference* (p. 1) is intended to provide information about individual commands. Although the reference also contains lists of commands arranged by functional grouping (p. 11), it is not intended to be your primary source of procedural information. For introductory information and procedural guidelines concerning when and where to use commands, see the appropriate analysis guide for your application.

A command mentioned in the various analysis guides implies a link to the detailed command description given in this reference. For ordering purposes, the alphabetical ordering of commands that begin with a star (*) or a slash (/) ignores those symbols.

A printed copy of the *Commands Quick Reference Guide* is available on the Ansys customer site in the documentation area.

The following *Command Reference* (p. 1) topics are available:

1.1. Conventions Used in This Reference

1.2. Terminology

1.3. Command Characteristics

1.1. Conventions Used in This Reference

This reference uses the following conventions to help you identify various types of information:

Type style or text	Indicates
UPPERCASE or **UPPERCASE**	Uppercase indicates an element name (such as SOLID273 or ELBOW290). Bold uppercase indicates a command name (such as **K (p. 969)** or **DDELE (p. 505)**).
Bold> Bold	Bold text in mixed case indicates a menu path within the graphical user interface (GUI), a series of menu options for accessing a command from the GUI. One or more angle brackets (>) separate menu items in a menu path. A command shown in the text is sometimes followed by its GUI equivalent in parentheses, as shown in this example:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type style or text	Indicates
	***GET (p. 797)** command (**Utility Menu> Parameters> Get Scalar Data**).
ITALICS	Uppercase italic letters (such as *VALUE*, *INC*, *TIME*) generally indicate command arguments for *numeric* values. In some cases (where indicated), non-numeric convenience labels (such as ON, OFF and ALL) can also be specified for these arguments.
Italics	Mixed case italic letters (such as *Lab* or *Fname*) generally indicate command arguments for *alphanumeric* values.
COURIER	A courier font indicates command input or output.

1.1.1. Applicable Products

This document applies to the following family of products:

Ansys Mechanical Pro (structural, full thermal, and vibration analysis)

Ansys Mechanical Premium (nonlinear structural, full thermal, vibration, linear dynamics analysis)

Ansys Mechanical Enterprise (advanced nonlinear structural, full thermal, vibration, linear dynamics and nonlinear transient dynamics, fracture, acoustics, full coupled-field, hydrodynamics)

Ansys Mechanical Enterprise PrepPost

Ansys Mechanical Enterprise Solver (batch mode)

Note:

While connection capabilities and High Performance Computing are included as part of the release distribution, they are separately licensed and separately installed products. Contact your Support Representative if you want to install and run any of these separately licensed products at your site.

1.1.2. Valid Products

Some commands are only valid in a subset of the Applicable Products (p. 2). The valid products for each command appear (as abbreviations) in the **Valid Products** list, below and to the right of each short command description.

The abbreviations correspond to the products as shown in this table:

Product	Abbreviation
Ansys Mechanical Pro	Pro
Ansys Mechanical Premium	Premium
Ansys Mechanical Enterprise	Enterprise

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Product	Abbreviation
Ansys Mechanical Enterprise PrepPost	PrepPost
Ansys Mechanical Enterprise Solver	Solver
Additive Suite add-on (requires one of the Enterprise products)	AS add-on

If the abbreviation in the **Valid Products** list is dark bold, the command is valid in that product. If the abbreviation is light gray ("grayed out"), the command is not valid or applicable and should not be used in that product. For example,

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | PrepPost | **Solver** | AS add-on

indicates the command is valid in Ansys Mechanical Pro, Ansys Mechanical Premium, Ansys Mechanical Enterprise, and Ansys Mechanical Enterprise Solver, but not in Ansys Mechanical Enterprise PrepPost. The "AS add-on" code behaves a bit differently. When this code is bold, the command is *only* available in the Additive Suite, an add-on to any of the Enterprise products.

Even if a command is shown as valid in a particular product, some options on that command may be invalid in that product. Any invalid options on an otherwise valid command are documented in the **Product Restrictions** section of that command documentation.

1.2. Terminology

Various terms are used in the command descriptions throughout this reference, defined as follows:

Analysis -- The set of input lines relating to a single problem. An analysis basically consists of three phases: the Preprocessing Phase, the Solution Phase, and the Postprocessing Phase.

Argument Default -- The value assumed for a command argument if the argument is not specified. See Command and Argument Defaults (p. 7).

Command -- An instruction to supply data, or control, to the program. Commands usually begin with a prescribed name, followed by alphanumeric data. For example, the command **ET (p. 686)**, *ITYPE*, *Ename*, ... may be input as **ET (p. 686)**,1,PIPE288 stating that element type 1 is defined as the PIPE288 element. The uppercase argument name indicates that a numerical value is typically entered in that field, whereas an upper-lower case data label indicates that an alphanumeric value is typically entered in that field. Another command, for example, **/PREP7 (p. 1479)**, instructs the program to enter the PREP7 portion of the program. All valid commands are alphabetically listed in the Command Dictionary (p. 65) of this document. Unrecognized commands are further processed in a macro search (***USE (p. 2059)**) before being ignored. Commands may be indented on a line for clarity.

Command Default -- Default program settings when a command is *not* issued. Default settings associated with a command are listed in the "Command Default" section of command description. See Command and Argument Defaults (p. 7). Not all commands have associated default settings (for example, action commands such as **SOLVE (p. 1822)** and **PRNSOL (p. 1505)**).

Data -- Data may be numeric (real or integer), alphabetic, or alphanumeric (containing letters and numbers). Nonnumeric data should not contain special characters such as

```
!   @   #    $   %    ^   &   *   (     )   _    -   +
=   |   \   {   }   [   ]    "     '   /    <     >   ~    `
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Numeric data may be input in a variety of ways as described in Data Input (p. 5). Some commands are switches of the form **Commandname**,*Key* where *Key* can be 0, NO or OFF to toggle the switch off; or 1, YES or ON to toggle the switch on.

Degree of Freedom -- The degrees of freedom are the primary nodal unknowns determined by the analysis. They may be displacements, rotations, temperatures, pressures, voltages, etc. A degree of freedom is defined by a node number and a label, for example, 1 UX, 87 ROTZ, 4 TEMP, etc. Derived results, such as stresses, heat flows, etc. are computed from the degree of freedom results and are considered secondary unknowns.

Element types with unequal sets of degrees of freedom can be combined in a single structure. For example, a 2D structure composed of 2D solid elements having two degrees of freedom (UX, UY) at each node and a 2D beam element having three degrees of freedom (UX, UY, ROTZ) at each node will have the latter three degrees of freedom at the common nodes. Nodes which do not have a beam element attached will have only two degrees of freedom with ROTZ eliminated from the solution printout.

Field -- The command name and data items entered on a command are separated into consecutive fields. A field is assumed to be as "wide" as the number of characters specified. A comma is used to end one field and begin the next.

Line -- A line of input is a physical record read by the computer. Input lines are limited to 640 characters (including preceding blanks, commas, and any special characters). For file input, a line is represented by one 640 column data record. For interactive input, a line is the string of characters (640 maximum) entered before the RETURN key is pressed. Several commands may be put on one line as described in Data Input (p. 5). Blank lines are permitted for clarity.

Command name -- Only the first four characters of any alphabetic (or alphanumeric) command name are interpreted by the program (except as noted for certain commands, such as **/POST*N***, **/AUX*N***. ***END-DO (p. 636)**, etc.). The remaining characters of the field are ignored. Names shown with fewer than four characters are assumed to have blanks up through the fourth character as part of the name. For example, this sample **ET (p. 686)** command may be input as **ET (p. 686)**,1,42 or **ET (p. 686)** ,1,42 or **ET (p. 686)** ,1,42, but not as **ETABCD**,1,42. Names may not contain special characters (as described in Data Input (p. 5)). If the command name is omitted, the name defaults to the name of the previous command, unless it was a slash (/) or star (*) command.

Postprocessing Phase -- The set of commands causing further processing of the solution output. These commands consist of commands from the POST1 and POST26 processors. The postprocessing phase may consist of input for several postprocessing sessions (in series).

Preprocessing Phase -- The set of commands related to defining the model. The preprocessing phase consists of input from the PREP7 processor.

Program -- The collection of all processors (preprocessing, postprocessing, auxiliary, etc.) is called the Mechanical APDL program.

Processor -- A group of related functions, such as model definition (PREP7) or results examination (POST1).

Run -- The collection of all commands between the system-level commands is called a run or session). A session may also consist of several analyses in series (separated by a **/CLEAR (p. 350)** command).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Phase -- The set of commands which apply boundary conditions to the model created in preprocessing, then performs a solution for that set of boundary conditions. The solution phase may consist of several solutions in series, such as a static solution, followed by a modal solution, etc.

1.3. Command Characteristics

The following characteristics apply to commands:

1.3.1. Data Input
1.3.2. Free-Format Input
1.3.3. Nonrestrictive Data Input
1.3.4. Condensed Data Input
1.3.5. Units
1.3.6. Command and Argument Defaults
1.3.7. File Names
1.3.8. Star and Slash Commands

1.3.1. Data Input

The data input for each command is described in this manual. Data should not be input in any undocumented field, nor should other than documented values be input in any field. Also, the data input described in this manual should not be used with any earlier version of the program. Some features that allow easy input of data are free-format, nonrestrictive, and condensed input.

1.3.2. Free-Format Input

Free-format capability allows the user to input data in consecutive fields without having to space to each field. The comma (,) character effectively ends the field so that the next character will be input in the beginning of the next field. A blank field is skipped by having no data item between the commas. Fields are assumed to be as wide as the number of characters specified. Input is converted to formatted fields when coded files are written (of width large enough (16 characters maximum) to minimize loss of accuracy).

Significant figures of output should not be expected to match that of input. Machine precision, rounding of numbers when writing internal scratch files, etc., tend to lower the precision during the analysis.

1.3.3. Nonrestrictive Data Input

Nonrestrictive data input allows the user to enter any form of data in a field and the Mechanical APDL program will interpret it as required (integer, real, or alphabetic) or ignore it. Double precision is used throughout the program for all real numbers. Alphabetic data may be entered in upper or lower case. Lower case is internally converted to upper case within the program (except for case-sensitive applications, such as in comments, (text preceded by a !) titles, and file names.)

The following features are available with the nonrestrictive data input.

- No distinction is necessary between real and integer data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Data may be placed anywhere within the field.
- Real data input without a decimal point has the decimal point assumed after the right-most digit.
- Real number values input to integer data fields will be rounded to the nearest integer. The absolute value of integer data must fall between zero and 2,000,000,000.
- Element and node numbers (IDs) should be between 1 and 999,999,999. Element type, material reference, real constant, section, and coordinate system IDs should be between 1 and 9,999,999. IDs larger than these may not be written out correctly in listings (xLIST commands), **CDWRITE (p. 293)** files, **LSWRITE (p. 1087)** files, and other text-based output files.
- Exponents may be input in the field after the number. The E (or D) character must be used (upper or lower case). The sign after the E character is assumed to be the sign associated with the exponent (the absence of a sign is assumed to be +). The acceptable range of values for real data is +/-1.0E+200 to +/-1.0E-200.
- A nonnumeric character in the numeric field (other than a valid convenience label, a parameter name, or an E or D exponent character) will be ignored and will cause the remainder of the field to be ignored. When the program can accept an alphanumeric label in a field, it will interpret any alphanumeric input that matches a valid label as the appropriate label. In other words, the program will not interpret a valid label as an identically named parameter. If you truly want to use a parameter in place of a valid label, you can use forced parametric substitution (using % signs). A non-alphanumeric character in a label field is taken as part of the label. Non-alphanumeric characters (such as CONTROL-characters, TAB-characters, and other terminal editing characters) should be avoided since they may be used directly instead of being interpreted by the terminal.

1.3.4. Condensed Data Input

An option for condensing the data input is available with the $ character. This character effectively ends a command and allows the next command to begin on the same line (at the next column). The line must not extend beyond 640 characters (including all commas and $ signs). If a command cannot be completed on the line, it should begin on the next line. Blank commands are ignored. The $ character should not be used following any command that causes a file switch to read additional commands (such as the **/INPUT (p. 948)**, ***USE (p. 2059)**, **CDREAD (p. 291)**, etc. commands, any "unknown command" macro, or any do-loops or if-then-else constructs). The $ character should not be used between a command and its required following format line (such as ***VREAD (p. 2153)**, ***VWRITE (p. 2174)**, ***MSG (p. 1185)**, etc.). The $ character should not be used after the **ALLSEL (p. 114)** command. Using the $ character in interactive mode may result in unexpected output behavior.

1.3.5. Units

The Mechanical APDL program permits the use of any consistent set of units for length, force, time, temperature, etc. Care must be taken to ensure that all input data are in the same set of units. The **/UNITS (p. 2054)** command may be used to note the system of units being used. Temperatures may be absolute or relative in most cases. For problems requiring absolute temperatures, such as those involving creep, swelling, or radiation, temperatures may be input as Celsius or Fahrenheit, for convenience, with a temperature shift from absolute zero (**TOFFST (p. 2008)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3.6. Command and Argument Defaults

To minimize the amount of data input, most commands have defaults which apply both to the command itself and to any arguments.

Command Defaults

A *command default* refers to the specification or behavior that is assumed when a command is not issued. For example, if you do not issue the **/FILNAME (p. 757)** command, the jobname defaults to FILE (or whatever jobname was specified when you entered the program).

Argument Defaults

An *argument default* is the value assumed for a command argument if the argument is not specified. For example, if you issue the command **N (p. 1207)**,10 (defining node 10 with the *X, Y, Z* coordinate arguments left blank), the node is defined at the origin; that is, *X, Y*, and *Z* default to zero.

Values for numeric arguments (such as *X, Y, Z*) typically default to zero, although exceptions exist for some commands, such as those with arguments specifying coefficients or tolerances. Such exceptions are noted in the documentation for those commands.

When an argument has a nonzero default, entering a blank *or a zero* for that argument triggers the nonzero default value. If an effective zero is desired for such an argument, input a very small number (such as 1E-14) instead.

In cases where no default is indicated or obvious for an argument, enter the desired value. Such arguments have no favored default value and require an explicit input value.

Some Command Defaults May Differ According to Product

The defaults for some commands and their arguments may differ depending on which product is using the commands. The "Product Restrictions" section of the descriptions of the affected commands clearly documents such cases.

If you intend to use your input file in more than one product, it is good practice to explicitly specify commands and argument values, rather than letting them default. Otherwise, behavior in the other product may differ from what you expect.

1.3.7. File Names

Various files are used during a session for reading, writing, and storing data. Program-generated files are identified by a name, a system-dependent separator, and an extension (as in the form `fname.ext`).

The name is obtained from the `Jobname` (250 characters maximum, including the directory name) as defined with the execution command. (Windows cannot create directory names longer than 245 characters.)

Unless otherwise specified, the `Jobname` defaults to a system-dependent name (usually `FILE`). The extension is a two- to four-character identifier. (See Program-Generated Files in the *Basic Analysis Guide*.) For example, the error message file (which has the identifier ERR) with a jobname of `FILE` and separator (.) would be generated as `FILE.ERR`. If the Jobname is `JOB1`, the file would be named `JOB1.ERR`. The documentation generically refers to program-generated files as `Jobname.identi-`

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fier, such as Jobname.ERR for the error message file. Unless otherwise indicated, files are written in the current working directory.

Some commands (such as **/INPUT (p. 948)** and **/OUTPUT (p. 1334)**) allow you to specify a file name other than Jobname, and an extension other than the default extension. Such user-written files use a naming convention similar to the Mechanical APDL-generated files except that the name (248 characters maximum, including the directory name) is user defined, and the extension (8 characters maximum) is optional. The file name can include the directory path, but it is not necessary if you want to use the default directory. File names containing directory paths must reference existing directories. File and directory names may contain blank spaces, but strings containing blank spaces must be enclosed in single quotes.

A system dependent separator is automatically written between file name and extension (if the extension exists) input on commands.

File name fields are usually restricted to 248 characters (including the directory name) in command inputs. If you specify no directory path, the default path is your working directory, and you can use all 248 characters for the file name.

To help ensure portability of input files across systems, the program has certain requirements for file names. Avoid using special characters (+,-,*,/,$,!, etc.) in file names unless meaningful, and begin file names with a letter. Some system-dependent special characters (such as ~) are not interpreted the same as they are at the operating system level when used in path names. Parameter substitution can be forced within the name, extension, or directory fields by enclosing the parameter within percent (%) signs. For example, if I = 5, the name TEST%I% becomes TEST5.

Chinese and Japanese character file names are supported. The input file, output file, Jobname, and working directory can include Chinese or Japanese characters in the file or folder name. These names can be input at the command line, through the Mechanical APDL Launcher, or through commands such as **/FILNAME (p. 757)**, **RESUME (p. 1601)**, and **/ASSIGN (p. 201)**. Keep the following points in mind when using Chinese or Japanese characters:

- The locale setting of the operating system must match that of the language being used for naming.
- The file extensions must remain in standard English characters.
- Non-English characters are not supported by GUI operations. Specifically, the **Open File** button on the Standard Toolbar does not accept Mechanical APDL files that are named with non-English characters, and the GUI command input area does not accept commands that include non-English characters.

1.3.8. Star and Slash Commands

The slash (/) and star (*) commands are usually used for supplying general control instructions to the program. For example, slash commands are used for entering a processor (**/PREP7 (p. 1479)**, **/POST1 (p. 1461)**, etc.). Printout controls, display controls, and file controls are available within the slash command set.

Repeat, looping and macro controls are available within the star command set.

Many star and slash commands are global and apply to all processors. For example, the **/TITLE (p. 2007)** command may be used to change the title in any processor (PREP7, POST1, etc.). The star commands

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can be input anywhere. Most slash commands can be input anywhere; however, some can only be input at a Begin level, and those commands are specifically noted in the command description. Graphics displays are controlled via the graphics display slash commands. The graphics display slash commands can be input anywhere.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Command Groupings

Following are tables of related commands. The documentation for each command also references these tables, providing convenient access to related commands.

> **Example:** To define material types in the PREP7 general preprocessor, see PREP7 Commands (p. 22) and examine the commands available under Table 2.28: Materials (p. 24).

While this document is useful as a reference, you should first look at the appropriate analysis guide to learn which commands are used for a given operation or analysis type.

The following command groupings are available:

2.1. SESSION Commands
2.2. DATABASE Commands
2.3. GRAPHICS Commands
2.4. APDL Commands
2.5. PREP7 Commands
2.6. SOLUTION Commands
2.7. POST1 Commands
2.8. POST26 Commands
2.9. AUX2 Commands
2.10. AUX3 Commands
2.11. AUX12 Commands
2.12. AUX15 Commands
2.13. Mapping Processor Commands

2.1. SESSION Commands

These commands provide general control to the session. The commands are grouped by functionality.

Table 2.1: Run Controls

These SESSION commands control the overall characteristics of the session, including the jobname, Graphical User Interface behavior, and file switching.	
/BATCH (p. 217)	Sets the program mode to "batch."
/CONFIG (p. 410)	Assigns values to configuration parameters.
/CWD (p. 452)	Changes the current working directory.
/EOF (p. 645)	Exits the file being read.
/EXIT (p. 708)	Stops the run and returns control to the system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SESSION commands control the overall characteristics of the session, including the jobname, Graphical User Interface behavior, and file switching.

/FILNAME (p. 757) Changes the jobname for the analysis.

HELP (p. 903) Displays help information for commands and element types.

/INPUT (p. 948) Switches the input file for the commands that follow.

KEYW (p. 981) Sets a keyword used by the GUI for context filtering (GUI).

MEMM (p. 1120) Enables the current session to keep allocated memory.

/MENU (p. 1121) Activates the Graphical User Interface (GUI).

/MSTART (p. 1198) Controls the initial GUI components.

/NERR (p. 1215) Limits the number of warning and error messages displayed.

/OUTPUT (p. 1334) Redirects text output to a file or to the screen.

PAUSE (p. 1356) Temporarily releases (pauses) the currently used product license so that another application can use it.

/STATUS (p. 1865) Lists the status of items for the run.

/SYP (p. 1894) Passes a command string and arguments to the operating system.

/SYS (p. 1895) Passes a command string to the operating system.

/UI (p. 2040) Activates specified GUI dialog boxes.

/UIS (p. 2045) Controls the GUI behavior.

UNPAUSE (p. 2056) Restores use of a temporarily released (paused) product license. (Applicable only *after* a previously issued **PAUSE (p. 1356)** command.)

Table 2.2: Processor Entry

These SESSION commands enter and exit the various processors in the program.

/AUX2 (p. 208) Enters the binary file dumping processor.

/AUX3 (p. 209) Enters the results file editing processor.

/AUX12 (p. 209) Enters the radiation matrix generation processor.

/AUX15 (p. 210) Enters the IGES file transfer processor.

FINISH (p. 758) Exits normally from a processor.

/MAP (p. 1107) Enters the mapping processor.

/POST1 (p. 1461) Enters the database results postprocessor.

/POST26 (p. 1462) Enters the time-history results postprocessor.

/PREP7 (p. 1479) Enters the model creation preprocessor.

/QUIT (p. 1563) Exits a processor.

/SOLU (p. 1821) Enters the solution processor.

Table 2.3: Files

These SESSION commands control file operations, such as deleting, copying, and listing.

ANSTOAQWA (p. 153) Creates an AQWA-LINE input file from the current model.

ANSTOASAS (p. 156) Creates an ASAS input file from the current model.

/ASSIGN (p. 201) Reassigns a file name to a file identifier.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SESSION commands control file operations, such as deleting, copying, and listing.

/CLOG (p. 355)	Copies the session log file to a named file.
/COPY (p. 416)	Copies a file.
/DELETE (p. 514)	Deletes a file.
/FCLEAN (p. 736)	Deletes all local files in all processors in a distributed-memory parallel processing run.
/FCOMP (p. 737)	Specifies file compression level.
/FDELE (p. 744)	Deletes a binary file after it is used.
LGWRITE (p. 1048)	Writes the database command log to a file.
***LIST (p. 1055)**	Displays the contents of an external, coded file.
/RENAME (p. 1585)	Renames a file.
/UPF (p. 2050)	Links user-programmable routines into Mechanical APDL.

Table 2.4: List Controls

These SESSION commands control listings and printed program output.

C* (p. 283)**	Places a comment in the output.
/COM (p. 404)	Places a comment in the output.
/GO (p. 865)	Reactivates suppressed printout.
/GOLIST (p. 865)	Reactivates the suppressed data input listing.
/GOPR (p. 865)	Reactivates suppressed printout.
/NOLIST (p. 1249)	Suppresses the data input listing.
/NOPR (p. 1251)	Suppresses the expanded interpreted input data listing.

2.2. DATABASE Commands

These commands operate on the database in a global sense. The commands are grouped by functionality.

Table 2.5: Setup

These DATABASE commands initialize the database, save it to a file, or annotate it with titles and systems of units.

/CLEAR (p. 350)	Clears the database.
RESUME (p. 1601)	Resumes the database from the database file.
SAVE (p. 1645)	Saves all current database information.
/SMBC (p. 1806)	Controls the display of solid model boundary condition symbols and labels.
STAT (p. 1861)	Displays the status of database settings.
/STITLE (p. 1867)	Defines subtitles.
/TITLE (p. 2007)	Defines a main title.
UNDO (p. 2053)	Enables modifying or saving commands issued since the last **RESUME (p. 1601)** or **SAVE (p. 1645)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These DATABASE commands initialize the database, save it to a file, or annotate it with titles and systems of units.

/UNITS (p. 2054) Annotates the database with the system of units used.

Table 2.6: Selecting

These DATABASE commands select subsets of database entities for further operations.

ALLSEL (p. 114) Selects all entities with a single command.

ASLL (p. 199) Selects those areas containing the selected lines.

ASEL (p. 192) Selects a subset of areas.

ASLV (p. 200) Selects those areas contained in the selected volumes.

DOFSEL (p. 569) Selects a degree-of-freedom label set for reference by other commands.

ESEL (p. 661) Selects a subset of elements.

ESLA (p. 668) Selects those elements associated with the selected areas.

ESLL (p. 669) Selects those elements associated with the selected lines.

ESLN (p. 670) Selects those elements attached to the selected nodes.

ESLV (p. 671) Selects elements associated with the selected volumes.

KSEL (p. 995) Selects a subset of keypoints or hard points.

KSLL (p. 998) Selects those keypoints contained in the selected lines.

KSLN (p. 998) Selects those keypoints associated with the selected nodes.

LSEL (p. 1074) Selects a subset of lines.

LSLA (p. 1078) Selects those lines contained in the selected areas.

LSLK (p. 1079) Selects those lines containing the selected keypoints.

NSEL (p. 1266) Selects a subset of nodes.

NSLA (p. 1272) Selects those nodes associated with the selected areas.

NSLE (p. 1273) Selects those nodes attached to the selected elements.

NSLK (p. 1274) Selects those nodes associated with the selected keypoints.

NSLL (p. 1275) Selects those nodes associated with the selected lines.

NSLV (p. 1276) Selects those nodes associated with the selected volumes.

VSEL (p. 2163) Selects a subset of volumes.

VSLA (p. 2166) Selects those volumes containing the selected areas.

Table 2.7: Components

These DATABASE commands enable selected subsets of entities to be named as components for easy selection later on.

CM (p. 356) Groups geometry items into a component.

CMDELE (p. 362) Deletes a component or assembly definition.

CMEDIT (p. 365) Edits an existing component or assembly.

CMGRP (p. 365) Groups components and assemblies into an assembly.

CMLIST (p. 366) Lists the entities contained in a component or assembly.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These DATABASE commands enable selected subsets of entities to be named as components for easy selection later on.

CMMOD (p. 368) Modifies the specification of a component.

CMPLOT (p. 371) Plots the entities contained in a component or assembly.

CMSEL (p. 374) Selects a subset of components and assemblies.

CMWRITE (p. 381) Writes components and assemblies to a file.

Table 2.8: Working Plane

These DATABASE commands activate, move, rotate, and modify the working plane, which is used for picking operations.

KWPAVE (p. 1003) Moves the working plane origin to the average location of keypoints.

KWPLAN (p. 1004) Defines the working plane using three keypoints.

LWPLAN (p. 1095) Defines the working plane normal to a location on a line.

NWPAVE (p. 1303) Moves the working plane origin to the average location of nodes.

NWPLAN (p. 1304) Defines the working plane using three nodes.

WPAVE (p. 2178) Moves the working plane origin to the average of specified points.

WPCSYS (p. 2179) Defines the working plane location based on a coordinate system.

WPLANE (p. 2180) Defines a working plane to assist in picking operations.

WPOFFS (p. 2181) Offsets the working plane.

WPROTA (p. 2181) Rotates the working plane.

WPSTYL (p. 2182) Controls the display and style of the working plane.

Table 2.9: Coordinate System

These DATABASE commands define and manipulate coordinate systems.

CLOCAL (p. 352) Defines a local coordinate system relative to the active coordinate system.

CS (p. 435) Defines a local coordinate system by three node locations.

CSCIR (p. 436) Locates the singularity for non-Cartesian local coordinate systems.

CSDELE (p. 438) Deletes local coordinate systems.

CSKP (p. 438) Defines a local coordinate system by three keypoint locations.

CSLIST (p. 440) Lists coordinate systems.

CSWPLA (p. 440) Defines a local coordinate system at the origin of the working plane.

CSYS (p. 441) Activates a previously defined coordinate system.

LOCAL (p. 1059) Defines a local coordinate system by a location and orientation.

Table 2.10: Picking

These DATABASE commands are generated by the GUI when picking operations are performed.

FITEM (p. 758) Identifies items chosen by a picking operation (GUI).

FLST (p. 766) Specifies data required for a picking operation (GUI).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3. GRAPHICS Commands

These commands control Mechanical APDL graphics. The commands are grouped by functionality.

Table 2.11: Setup

These GRAPHICS commands are used for general graphics control, such as window setup, color settings, specifications saving, etc.

/CMAP (p. 359)	Changes an existing color mapping table or creates a new one .
/COLOR (p. 400)	Specifies the color mapping for various items.
/DEVICE (p. 523)	Controls graphics device options.
DSYS (p. 582)	Activates a display coordinate system for geometry listings and plots.
/DV3D (p. 584)	Sets 3D device option modes.
/ERASE (p. 653)	Specifies that the screen is to be erased before each display.
ERASE (p. 652)	Explicitly erases the current display.
/GCMD (p. 795)	Controls the type of element or graph display used for the **GPLOT (p. 866)** command.
/GCOLUMN (p. 795)	Applies a label to a specified curve.
/GFILE (p. 858)	Specifies the pixel resolution on Z-buffered graphics files.
GPLOT (p. 866)	Controls general plotting.
/GRAPHICS (p. 866)	Defines type of graphics display to be used for element plots.
/GRESUME (p. 868)	Sets graphics settings to the settings on a file.
/GSAVE (p. 875)	Saves graphics settings to a file for later use.
/GTYPE (p. 883)	Controls the entities that the **GPLOT (p. 866)** command displays.
/IMAGE (p. 933)	Enables graphics data to be captured and saved.
IMMED (p. 937)	Enables immediate display of a model as it is generated.
JPEG (p. 965)	Provides JPEG File Export for displays.
/MREP (p. 1182)	Enables you to reissue the graphics command macro "name" during a replot or zoom operation.
/NOERASE (p. 1249)	Prevents the screen erase between displays.
PNGR (p. 1455)	Sets file options for PNGR graphics export for displays.
/PSTATUS (p. 1550)	Displays the global or window display specifications.
/REPLOT (p. 1587)	Automatically reissues the last display command for convenience.
/RESET (p. 1599)	Resets display specifications to their initial defaults.
/SEG (p. 1707)	Enables graphics data to be stored in the local terminal memory.
/SHOW (p. 1785)	Specifies the device and other parameters for graphics displays.
TIFF (p. 1999)	Provides TIFF file Export for displays.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These GRAPHICS commands are used for general graphics control, such as window setup, color settings, specifications saving, etc.

/WINDOW (p. 2177) Defines the window size on the screen.

Table 2.12: Views

These GRAPHICS commands control the view of the model.

/ANGLE (p. 140) Rotates the display about an axis.

/AUTO (p. 207) Resets the focus and distance specifications to "automatically calculated."

/DIST (p. 536) Specifies the viewing distance for magnifications and perspective.

/FOCUS (p. 771) Specifies the focus point (center of the window).

/USER (p. 2061) Conveniently resets **/FOCUS (p. 771)** and **/DIST (p. 536)** to USER.

/VCONE (p. 2078) Defines the view cone angle for perspective displays.

/VIEW (p. 2131) Defines the viewing direction for the display.

/VUP (p. 2173) Specifies the global Cartesian coordinate system reference orientation.

/XFRM (p. 2194) Controls the centroid or the axis of dynamic rotation.

/ZOOM (p. 2203) Zooms a region of a display window.

Table 2.13: Scaling

These GRAPHICS commands scale various parts of the display.

/DSCALE (p. 575) Sets the displacement multiplier for displacement displays.

/ICLWID (p. 924) Scales the line width of circuit builder icons.

/ICSCALE (p. 927) Scales the icon size for elements supported in the circuit builder.

/RATIO (p. 1572) Distorts the object geometry.

/SHRINK (p. 1794) Shrinks elements, lines, areas, and volumes for display clarity.

/SSCALE (p. 1847) Sets the contour multiplier for topographic displays.

/TXTRE (p. 2029) Applies textured appearance to selected items.

/VSCALE (p. 2160) Scales the length of displayed vectors.

Table 2.14: Style

These GRAPHICS commands change the way a model is displayed.

/CPLANE (p. 426) Specifies the cutting plane for section and capped displays.

/CTYPE (p. 443) Specifies the type of contour display.

/EDGE (p. 594) Displays only the "edges" of an object.

/ESHAPE (p. 665) Displays elements with shapes determined from the real constants.

/FACET (p. 731) Specifies the facet representation used to form solid model displays.

/GLINE (p. 860) Specifies the element outline style.

/GMARKER (p. 861) Specifies the curve marking style.

GMFACE (p. 863) Specifies the facet representation used to form solid models.

/LIGHT (p. 1049) Specifies the light direction for the display window.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These GRAPHICS commands change the way a model is displayed.

/NORMAL (p. 1253) Enables displaying area elements by top or bottom faces.

/SHADE (p. 1783) Defines the type of surface shading used with Z-buffering.

/TRLCY (p. 2015) Specifies the level of translucency.

/TYPE (p. 2034) Defines the type of display.

Table 2.15: Labeling

These GRAPHICS commands add helpful labels and symbols to displays.

/CFORMAT (p. 317) Controls the graphical display of alpha character strings for parameters, components, assemblies, and tables.

/CLABEL (p. 349) Specifies contour labeling.

/CONTOUR (p. 415) Specifies the uniform contour values on stress displays.

/CVAL (p. 450) Specifies nonuniform contour values on stress displays.

/GFORMAT (p. 859) Specifies the format for the graphical display of numbers.

/HBC (p. 887) Determines how boundary conditions are displayed in a display window.

/NUMBER (p. 1290) Specifies whether numbers, colors, or both are used for displays.

/PBC (p. 1356) Shows boundary condition symbols and values on displays.

/PBF (p. 1360) Shows body-force loads as contours on displays.

/PLOPTS (p. 1435) Controls graphics options on subsequent displays.

/PNUM (p. 1457) Controls entity numbering/coloring on plots.

/PSF (p. 1537) Shows surface load symbols on model displays.

/PSYMB (p. 1552) Shows various symbols on displays.

/TRIAD (p. 2014) Shows the global XYZ coordinate triad on displays.

/UDOC (p. 2038) Determines position and content for the multi-legend options.

Table 2.16: Graphs

These GRAPHICS commands control the way line graphs are displayed.

/AXLAB (p. 216) Labels the X and Y axes on graph displays.

/GRID (p. 869) Selects the type of grid on graph displays.

/GROPT (p. 870) Sets various line graph display options.

/GRTYP (p. 874) Selects single or multiple Y-axes graph displays.

/GTHK (p. 882) Sets line thicknesses for graph lines.

/XRANGE (p. 2199) Specifies a linear abscissa (X) scale range.

/YRANGE (p. 2201) Specifies a linear ordinate (Y) scale range.

Table 2.17: Annotation

These GRAPHICS commands annotate a display with notes and symbols.

/AN3D (p. 129) Specifies 3D annotation functions

/ANNOT (p. 145) Activates graphics for annotating displays (GUI).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These GRAPHICS commands annotate a display with notes and symbols.

/ANUM (p. 166)	Specifies the annotation number, type, and hot spot (GUI).
/LARC (p. 1011)	Creates annotation arcs (GUI).
/LINE (p. 1051)	Creates annotation lines (GUI).
/LSPEC (p. 1081)	Specifies annotation line attributes (GUI).
/LSYMBOL (p. 1088)	Creates annotation symbols (GUI).
/PCIRCLE (p. 1370)	Creates an annotation circle (GUI).
/PMORE (p. 1454)	Creates an annotation polygon (GUI).
/POLYGON (p. 1460)	Creates annotation polygons (GUI).
/PSPEC (p. 1548)	Creates annotation polygon attributes (GUI).
/PWEDGE (p. 1557)	Creates an annotation wedge (GUI).
/TLABEL (p. 2007)	Creates annotation text (GUI).
/TSPEC (p. 2024)	Creates annotation text attributes (GUI).

2.4. APDL Commands

These commands make up the Ansys Parametric Design Language (APDL). The commands are grouped by functionality.

Table 2.18: Parameter Definition

These APDL commands define parameters and their values.

***AFUN (p. 83)**	Specifies units for angular functions in parameter expressions.
***ASK (p. 84)**	Prompts the user to input a parameter value.
***DEL (p. 512)**	Deletes a parameter (GUI).
***DIM (p. 530)**	Defines an array parameter and its dimensions.
***GET (p. 797)**	Retrieves a value and stores it as a user-named parameter.
/INQUIRE (p. 950)	Returns system information to a parameter.
PARRES (p. 1351)	Reads parameters from a file.
PARSAV (p. 1352)	Writes parameters to a file.
***SET (p. 1720)**	Assigns values to user-named parameters.
***STATUS (p. 1863)**	Lists the current parameters and abbreviations.
***TAXIS (p. 1898)**	Defines table index numbers.
***TREAD (p. 2012)**	Reads data from an external file into a table array parameter.
***VFILL (p. 2097)**	Fills an array parameter.
***VGET (p. 2118)**	Retrieves values into an array parameter.
***VREAD (p. 2153)**	Reads data and produces an array parameter vector or matrix.

Table 2.19: Macro Files

These APDL commands build and execute command macros.

***CFCLOS (p. 316)**	Closes the "command" file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These APDL commands build and execute command macros.

***CFOPEN (p. 316)**	Opens a "command" file.
***CFWRITE (p. 318)**	Writes a command (or similar string) to a "command" file.
***CREATE (p. 434)**	Opens (creates) a macro file.
/DFLAB (p. 527)	Changes degree of freedom labels for user custom elements.
***END (p. 635)**	Closes a macro file.
/MKDIR (p. 1129)	Creates a directory.
***MSG (p. 1185)**	Writes an output message via the message subroutine.
/PMACRO (p. 1449)	Specifies that macro contents be written to the session log file.
/PSEARCH (p. 1535)	Specifies a directory to be searched for "unknown command" macro files.
/RMDIR (p. 1612)	Removes (deletes) a directory.
/TEE (p. 1995)	Writes a list of commands to a specified file at the same time that the commands are being executed.
***ULIB (p. 2051)**	Identifies a macro library file.
***USE (p. 2059)**	Executes a macro file.

Table 2.20: Encryption/Decryption

These APDL commands encrypt and decrypt command input and other data.

/DBDECRYPT (p. 499)	Controls decryption of material data in the database file.
/DBENCRYPT (p. 500)	Controls encryption of material data in the database file.
/DECRYPT (p. 510)	Controls decryption of command input.
/ENCRYPT (p. 634)	Controls encryption of command input.

Table 2.21: Abbreviations

These APDL commands can be used to define abbreviations for longer commands, and to create user-defined commands.

***ABBR (p. 81)**	Defines an abbreviation.
ABBRES (p. 86)	Reads abbreviations from a coded file.
ABBSAV (p. 87)	Writes the current abbreviation set to a coded file.
/UCMD (p. 2037)	Assigns a user-defined command name.

Table 2.22: Array Parameters

These APDL commands operate on parameter arrays (that is, vectors and matrices).

/DIRECTORY (p. 534)	Put the file names in the current directory into a string parameter array.
***MFOURI (p. 1124)**	Calculates the coefficients for, or evaluates, a Fourier series.
***MFUN (p. 1125)**	Copies or transposes an array parameter matrix.
***MOPER (p. 1147)**	Performs matrix operations on array parameter matrices.
***MWRITE (p. 1202)**	Writes a matrix to a file in a formatted sequence.
***SREAD (p. 1844)**	Reads a file into a string array parameter.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These APDL commands operate on parameter arrays (that is, vectors and matrices).

***TOPER (p. 2009)** Operates on table parameters.
***VABS (p. 2072)** Applies the absolute value function to array parameters.
***VCOL (p. 2077)** Specifies the number of columns in matrix operations.
***VCUM (p. 2080)** Enables array parameter results to add to existing results.
***VEDIT (p. 2089)** Enables numerical array parameters to be graphically edited.
***VFACT (p. 2092)** Applies a scale factor to array parameters.
***VFUN (p. 2106)** Performs a function on a single array parameter.
***VITRP (p. 2134)** Forms an array parameter by interpolation of a table.
***VLEN (p. 2136)** Specifies the number of rows to be used in array parameter operations.
***VMASK (p. 2139)** Specifies an array parameter as a masking vector.
***VOPER (p. 2142)** Operates on two array parameters.
***VPLOT (p. 2146)** Graphs columns (vectors) of array parameters.
***VPUT (p. 2149)** Restores array parameter values into the database.
***VSCFUN (p. 2161)** Determines properties of an array parameter.
***VSTAT (p. 2167)** Lists the current specifications for the array parameters.
***VWRITE (p. 2174)** Writes data to a file in a formatted sequence.

Table 2.23: Process Controls

These APDL commands can be used to control the order in which other commands are processed.

***CYCLE (p. 462)** Bypasses commands within a do-loop.
***DO (p. 567)** Defines the beginning of a do-loop.
***DOWHILE (p. 573)** Loops repeatedly through the next ***ENDDO (p. 636)** command.
***ELSE (p. 620)** Separates the final if-then-else block.
***ELSEIF (p. 620)** Separates an intermediate if-then-else block.
***ENDDO (p. 636)** Ends a do-loop and starts the looping action.
***ENDIF (p. 636)** Ends an if-then-else.
***EXIT (p. 707)** Exits a do-loop.
***GO (p. 864)** Causes a specified line on the input file to be read next.
***IF (p. 927)** Conditionally causes commands to be read.
***REPEAT (p. 1586)** Repeats the previous command.
***RETURN (p. 1605)** Returns input stream to a higher level.
/WAIT (p. 2177) Introduces a delay before reading the next command.

Table 2.24: Matrix Operations (APDL Math)

These APDL Math commands can be used to create, manipulate, and solve matrices.

***AXPY (p. 86)** Performs the matrix operation M2= v*M1 + w*M2.
***COMP (p. 405)** Compresses the columns of a matrix using a specified algorithm.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These APDL Math commands can be used to create, manipulate, and solve matrices.

***DMAT (p. 551)**	Creates a dense matrix.
***DOT (p. 572)**	Computes the dot (or inner) product of two vectors.
***EIGEN (p. 605)**	Performs a modal solution with unsymmetric or damping matrices.
***EXPORT (p. 716)**	Exports a matrix to a file in the specified format.
***FFT (p. 747)**	Computes the fast Fourier transformation of a specified matrix or vector.
***FREE (p. 776)**	Deletes a matrix or a solver object and frees its memory allocation.
***HPROD (p. 907)**	Performs a Hadamard vector product (C = A B).
***INIT (p. 945)**	Initializes a vector or dense matrix.
***INQUIRE (p. 949)**	Retrieves properties of an existing APDL Math Object.
***ITENGINE (p. 963)**	Performs a solution using an iterative solver.
***KRON (p. 991)**	Computes the Kronecker product of two matrices/vectors, $M_3 = M_1 \otimes M_2$
***LSBAC (p. 1066)**	Performs the solve (forward/backward substitution) of a factorized linear system.
***LSENGINE (p. 1072)**	Creates a linear solver engine.
***LSFACTOR (p. 1078)**	Performs the numerical factorization of a linear solver system.
***MERGE (p. 1122)**	Merges two dense matrices or vectors into one.
***MULT (p. 1201)**	Performs the matrix multiplication M3 = M1(T1)*M2(T2).
***NRM (p. 1260)**	Computes the norm of the specified matrix or vector.
***PRINT (p. 1496)**	Prints the matrix values to a file.
***REMOVE (p. 1583)**	Suppresses rows or columns of a dense matrix or a vector.
***RENAME (p. 1584)**	Renames an existing vector or matrix.
***SCAL (p. 1647)**	Scales a vector or matrix by a constant
***SMAT (p. 1801)**	Creates a sparse matrix.
***SORT (p. 1822)**	Sorts the values of the specified vector.
***VEC (p. 2085)**	Creates a vector.
***XPL (p. 2196)**	Accesses content of a Mechanical APDL file.

2.5. PREP7 Commands

These commands create and set up the model. The commands are grouped by functionality.

Table 2.25: Database

These PREP7 commands read model data into the database, list out the database, and control the numbering of entities in the database.

AFLIST (p. 106)	Lists the current data in the database.
CDREAD (p. 291)	Reads a file of solid model and database information into the database.
CDWRITE (p. 293)	Writes geometry and load database items to a file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands read model data into the database, list out the database, and control the numbering of entities in the database.

CDOPT (p. 290) Specifies format to be used for archiving geometry.

CECHECK (p. 299) Check constraint equations and couplings for rigid body motions.

CHECK (p. 333) Checks current database items for completeness.

CNCHECK (p. 382) Provides and/or adjusts the initial status of contact pairs.

FC (p. 732) Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.

FCCHECK (p. 735) Checks both the strain and stress input criteria for all materials.

FCDELE (p. 736) Deletes previously defined failure criterion data for the given material.

FCLIST (p. 737) To list what the failure criteria is that you have input.

IGESOUT (p. 932) Writes solid model data to a file in IGES Version 5.1 format.

NOOFFSET (p. 1250) Prevents the **CDREAD (p. 291)** command from offsetting specified data items.

NUMCMP (p. 1291) Compresses the numbering of defined items.

NUMMRG (p. 1295) Merges coincident or equivalently defined items.

NUMOFF (p. 1299) Adds a number offset to defined items.

NUMSTR (p. 1301) Establishes starting numbers for automatically numbered items.

/PREP7 (p. 1479) Enters the model creation preprocessor.

Table 2.26: Element Type

These PREP7 commands define the type of elements to be used in the model.

DOF (p. 568) Adds degrees of freedom to the current degree-of-freedom set.

ELBOW (p. 616) Specifies degrees of freedom to be coupled for end release and applies section constraints to elbow elements.

ET (p. 686) Defines a local element type from the element library.

ETCHG (p. 699) Changes element types to their corresponding types.

ETCONTROL (p. 700) Controls the element technologies used in element formulation (for applicable elements).

ETDELE (p. 702) Deletes element types.

ETLIST (p. 702) Lists currently defined element types.

KEYOPT (p. 979) Sets element key options.

NSVR (p. 1289) Defines the number of variables for user-programmable element options.

Table 2.27: Real Constants

These PREP7 commands define the model real constants.

R (p. 1565) Defines the element real constants.

RDELE (p. 1577) Deletes real constant sets.

RLIST (p. 1611) Lists the real constant sets.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands define the model real constants.

RMODIF (p. 1613) Modifies real constant sets.

RMORE (p. 1614) Adds real constants to a set.

SETFGAP (p. 1729) Updates real constant table for squeeze film elements.

Table 2.28: Materials

These PREP7 commands define the linear material properties.

EMUNIT (p. 632) Specifies the system of units for magnetic field problems.

MP (p. 1160) Defines a linear material property.

MPAMOD (p. 1166) Modifies temperature-dependent coefficients of thermal expansion.

MPCHG (p. 1167) Changes the material number attribute of an element.

MPCOPY (p. 1167) Copies material model data from one material reference number to another.

MPDATA (p. 1168) Defines property data to be associated with the temperature table.

MPDELE (p. 1172) Deletes linear material properties.

MPDRES (p. 1173) Reassembles existing material data with the temperature table.

/MPLIB (p. 1173) Sets the default material library read and write paths.

MPLIST (p. 1175) Lists linear material properties.

MPPLOT (p. 1175) Plots linear material properties as a function of temperature.

MPREAD (p. 1176) Reads a file containing material properties.

MPTEMP (p. 1178) Defines a temperature table for material properties.

MPTGEN (p. 1179) Adds temperatures to the temperature table by generation.

MPTRES (p. 1180) Restores a temperature table previously defined.

MPWRITE (p. 1181) Writes linear material properties in the database to a file (if the Lib option is not specified) or writes both linear and nonlinear material properties (if Lib is specified) from the database to a file.

NSMASS (p. 1277) Applies nonstructural mass to selected elements.

TBFPLOT (p. 1968) Plots material curve-fitting data.

TBFT (p. 1969) Performs material curve-fitting operations on materials.

UIMP (p. 2044) Defines constant material properties (GUI).

Table 2.29: Material Data Tables

These PREP7 commands create and modify the material data tables (that is, to specify and define material models).

TB (p. 1899) Activates a material data table.

TBCOPY (p. 1959) Copies a data table from one material to another.

TBDATA (p. 1960) Defines data for the material data table.

TBDELE (p. 1961) Deletes previously defined material data tables.

TBEO (p. 1961) Sets special options or parameters for material data tables.

TBFIELD (p. 1966) Defines values of field variables for the material data tables.

TBIN (p. 1963) Sets parameters used for interpolation of the material data tables.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands create and modify the material data tables (that is, to specify and define material models).

TBLIST (p. 1988) Lists the material data tables.

TBMODIF (p. 1989) Modifies data for the material data table (GUI).

TBPLOT (p. 1990) Displays the material data table.

TBPT (p. 1992) Defines a point on a stress-strain or B-H curve.

TBTEMP (p. 1993) Defines a temperature for the material data table.

Table 2.30: Primitives

These PREP7 commands create primitive shapes for modeling.

BLC4 (p. 266) Creates a rectangular area or block volume by corner points.

BLC5 (p. 267) Creates a rectangular area or block volume by center and corner points.

BLOCK (p. 268) Creates a block volume based on working plane coordinates.

CON4 (p. 409) Creates a conical volume anywhere on the working plane.

CONE (p. 409) Creates a conical volume centered about the working plane origin.

CYL4 (p. 477) Creates a circular area or cylindrical volume anywhere on the working plane.

CYL5 (p. 478) Creates a circular area or cylindrical volume by end points.

CYLIND (p. 479) Creates a cylindrical volume centered about the working plane origin.

PCIRC (p. 1369) Creates a circular area centered about the working plane origin.

POLY (p. 1460) Creates a polygonal area based on working plane coordinate pairs.

PRI2 (p. 1495) Creates a polygonal area or a prism volume by vertices (GUI).

PRISM (p. 1497) Creates a prism volume based on working plane coordinate pairs.

PTXY (p. 1556) Defines coordinate pairs for use in polygons and prisms.

RECTNG (p. 1580) Creates a rectangular area anywhere on the working plane.

RPOLY (p. 1616) Creates a regular polygonal area centered about the working plane origin.

RPR4 (p. 1617) Creates a regular polygonal area or prism volume anywhere on the working plane.

RPRISM (p. 1618) Creates a regular prism volume centered about the working plane origin.

SPH4 (p. 1830) Creates a spherical volume anywhere on the working plane.

SPH5 (p. 1830) Creates a spherical volume by diameter end points.

SPHERE (p. 1831) Creates a spherical volume centered about the working plane origin.

TORUS (p. 2010) Creates a toroidal volume.

Table 2.31: Keypoints

These PREP7 commands create, modify, list, etc., keypoints.

GSUM (p. 881) Calculates and prints geometry items.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands create, modify, list, etc., keypoints.

K (p. 969)	Defines a keypoint.
KBETW (p. 972)	Creates a keypoint between two existing keypoints.
KCENTER (p. 973)	Creates a keypoint at the center of a circular arc defined by three locations.
KDELE (p. 976)	Deletes unmeshed keypoints.
KDIST (p. 976)	Calculates and lists the distance between two keypoints.
KFILL (p. 981)	Generates keypoints between two keypoints.
KGEN (p. 982)	Generates additional keypoints from a pattern of keypoints.
KL (p. 983)	Generates a keypoint at a specified location on an existing line.
KLIST (p. 984)	Lists the defined keypoints or hard points.
KMODIF (p. 985)	Modifies an existing keypoint.
KMOVE (p. 986)	Calculates and moves a keypoint to an intersection.
KNODE (p. 987)	Defines a keypoint at an existing node location.
KPLOT (p. 988)	Displays the selected keypoints.
KPSCALE (p. 988)	Generates a scaled set of (meshed) keypoints from a pattern of keypoints.
KSCALE (p. 993)	Generates a scaled pattern of keypoints from a given keypoint pattern.
KSUM (p. 999)	Calculates and prints geometry statistics of the selected keypoints.
KSYMM (p. 1000)	Generates a reflected set of keypoints.
KTRAN (p. 1001)	Transfers a pattern of keypoints to another coordinate system.
SOURCE (p. 1824)	Defines a default location for undefined nodes or keypoints.

Table 2.32: Hard Points

These PREP7 commands create, modify, list, etc., hard points.

HPTCREATE (p. 908)	Defines a hard point.
HPTDELETE (p. 909)	Deletes selected hard points.

Table 2.33: Lines

These PREP7 commands create, modify, list, etc., lines.

BSPLIN (p. 275)	Generates a single line from a spline fit to a series of keypoints.
CIRCLE (p. 344)	Generates circular arc lines.
GSUM (p. 881)	Calculates and prints geometry items.
L (p. 1005)	Defines a line between two keypoints.
L2ANG (p. 1006)	Generates a line at an angle with two existing lines.
L2TAN (p. 1007)	Generates a line tangent to two lines.
LANG (p. 1009)	Generates a straight line at an angle with a line.
LARC (p. 1010)	Defines a circular arc.
LAREA (p. 1012)	Generates the shortest line between two keypoints on an area.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands create, modify, list, etc., lines.

LCOMB (p. 1025) Combines adjacent lines into one line.
LDELE (p. 1033) Deletes unmeshed lines.
LDIV (p. 1034) Divides a single line into two or more lines.
LDRAG (p. 1035) Generates lines by sweeping a keypoint pattern along path.
LEXTND (p. 1043) Extends a line at one end by using its slope.
LFILLT (p. 1044) Generates a fillet line between two intersecting lines.
LGEN (p. 1046) Generates additional lines from a pattern of lines.
LLIST (p. 1056) Lists the defined lines.
LPLOT (p. 1061) Displays the selected lines.
LREVERSE (p. 1063) Reverses the normal of a line, regardless of its connectivity or mesh status.
LROTAT (p. 1064) Generates circular lines by rotating a keypoint pattern about an axis.
LSSCALE (p. 1084) Generates a scaled set of lines from a pattern of lines.
LSTR (p. 1086) Defines a straight line irrespective of the active coordinate system.
LSUM (p. 1086) Calculates and prints geometry statistics of the selected lines.
LSYMM (p. 1089) Generates lines from a line pattern by symmetry reflection.
LTAN (p. 1091) Generates a line at the end of, and tangent to, an existing line.
LTRAN (p. 1091) Transfers a pattern of lines to another coordinate system.
SPLINE (p. 1832) Generates a segmented spline through a series of keypoints.
SSLN (p. 1848) Selects and displays small lines in the model.

Table 2.34: Areas

These PREP7 commands create, modify, list, etc., areas.

A (p. 81) Defines an area by connecting keypoints.
AATT (p. 85) Associates element attributes with the selected, unmeshed areas.
ADELE (p. 97) Deletes unmeshed areas.
ADGL (p. 98) Lists keypoints of an area that lie on a parametric degeneracy.
ADRAG (p. 101) Generates areas by dragging a line pattern along a path.
AFILLT (p. 105) Generates a fillet at the intersection of two areas.
AGEN (p. 107) Generates additional areas from a pattern of areas.
AL (p. 112) Generates an area bounded by previously defined lines.
ALIST (p. 113) Lists the defined areas.
ANORM (p. 147) Reorients area normals.
AOFFST (p. 168) Generates an area, offset from a given area.
APLOT (p. 169) Displays the selected areas.
AREVERSE (p. 181) Reverses the normal of an area, regardless of its connectivity or mesh status.
AROTAT (p. 182) Generates cylindrical areas by rotating a line pattern about an axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands create, modify, list, etc., areas.

ARSCALE (p. 183)	Generates a scaled set of areas from a pattern of areas.
ARSYM (p. 184)	Generates areas from an area pattern by symmetry reflection.
ASKIN (p. 198)	Generates an area by "skinning" a surface through guiding lines.
ASUB (p. 202)	Generates an area using the shape of an existing area.
ASUM (p. 203)	Calculates and prints geometry statistics of the selected areas.
ATRAN (p. 205)	Transfers a pattern of areas to another coordinate system.
GSUM (p. 881)	Calculates and prints geometry items.
SPLOT (p. 1833)	Displays the selected areas and a faceted view of their underlying surfaces

Table 2.35: Volumes

These PREP7 commands create, modify, list, etc., volumes.

EXTOPT (p. 722)	Controls options relating to the generation of volume elements from area elements.
GSUM (p. 881)	Calculates and prints geometry items.
V (p. 2069)	Defines a volume through keypoints.
VA (p. 2072)	Generates a volume bounded by existing areas.
VDELE (p. 2082)	Deletes unmeshed volumes.
VDGL (p. 2082)	Lists keypoints of a volume that lie on a parametric degeneracy.
VDRAG (p. 2084)	Generates volumes by dragging an area pattern along a path.
VEXT (p. 2091)	Generates additional volumes by extruding areas.
VGEN (p. 2116)	Generates additional volumes from a pattern of volumes.
VLIST (p. 2137)	Lists the defined volumes.
VLSCALE (p. 2137)	Generates a scaled set of volumes from a pattern of volumes.
VOFFST (p. 2140)	Generates a volume, offset from a given area.
VPLOT (p. 2147)	Displays the selected volumes.
VROTAT (p. 2155)	Generates cylindrical volumes by rotating an area pattern about an axis.
VSUM (p. 2167)	Calculates and prints geometry statistics of the selected volumes.
VSYMM (p. 2169)	Generates volumes from a volume pattern by symmetry reflection.
VTRAN (p. 2171)	Transfers a pattern of volumes to another coordinate system.

Table 2.36: Booleans

These PREP7 commands perform Boolean operations on solid model entities.

AADD (p. 82)	Adds separate areas to create a single area.
AGLUE (p. 108)	Generates new areas by "gluing" areas.
AINA (p. 109)	Finds the intersection of areas.
AINP (p. 110)	Finds the pairwise intersection of areas.
AINV (p. 110)	Finds the intersection of an area with a volume.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands perform Boolean operations on solid model entities.

AOVLAP (p. 169)	Overlaps areas.
APTN (p. 174)	Partitions areas.
ASBA (p. 186)	Subtracts areas from areas.
ASBL (p. 187)	Subtracts lines from areas.
ASBV (p. 189)	Subtracts volumes from areas.
ASBW (p. 190)	Subtracts the intersection of the working plane from areas (divides areas).
BOPTN (p. 269)	Specifies Boolean operation options.
BTOL (p. 279)	Specifies the Boolean operation tolerances.
LCSL (p. 1030)	Divides intersecting lines at their point(s) of intersection.
LGLUE (p. 1047)	Generates new lines by "gluing" lines.
LINA (p. 1050)	Finds the intersection of a line with an area.
LINL (p. 1053)	Finds the common intersection of lines.
LINP (p. 1053)	Finds the pairwise intersection of lines.
LINV (p. 1054)	Finds the intersection of a line with a volume.
LOVLAP (p. 1060)	Overlaps lines.
LPTN (p. 1061)	Partitions lines.
LSBA (p. 1065)	Subtracts areas from lines.
LSBL (p. 1067)	Subtracts lines from lines.
LSBV (p. 1069)	Subtracts volumes from lines.
LSBW (p. 1070)	Subtracts the intersection of the working plane from lines (divides lines).
VADD (p. 2074)	Adds separate volumes to create a single volume.
VGLUE (p. 2130)	Generates new volumes by "gluing" volumes.
VINP (p. 2133)	Finds the pairwise intersection of volumes.
VINV (p. 2134)	Finds the intersection of volumes.
VOVLAP (p. 2146)	Overlaps volumes.
VPTN (p. 2148)	Partitions volumes.
VSBA (p. 2156)	Subtracts areas from volumes.
VSBV (p. 2157)	Subtracts volumes from volumes.
VSBW (p. 2159)	Subtracts intersection of the working plane from volumes (divides volumes).

Table 2.37: Meshing

These PREP7 commands mesh solid models with nodes and elements.

ACCAT (p. 90)	Concatenates multiple areas in preparation for mapped meshing.
ACLEAR (p. 93)	Deletes nodes and area elements associated with selected areas.
AESIZE (p. 104)	Specifies the element size to be meshed onto areas.
AMAP (p. 116)	Generates a 2D mapped mesh based on specified area corners.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands mesh solid models with nodes and elements.

AMESH (p. 121) Generates nodes and area elements within areas.

AREFINE (p. 178) Refines the mesh around specified areas.

CHKMSH (p. 334) Checks area and volume entities for previous meshes.

CLRMSHLN (p. 355) Clears meshed entities.

CPCYC (p. 423) Couples the two side faces of a cyclically symmetric model for loadings that are the same on every segment.

CZDEL (p. 479) Edits or clears cohesive zone sections.

CZMESH (p. 480) Creates and meshes a cohesive zone group of elements.

DESIZE (p. 519) Controls default element sizes.

EORIENT (p. 646) Reorients solid element normals.

EREFINE (p. 654) Refines the mesh around specified elements.

ESIZE (p. 668) Specifies the default number of line divisions.

ESYS (p. 685) Sets the element coordinate system attribute pointer.

FVMESH (p. 784) Generates nodes and tetrahedral volume elements from detached exterior area elements (facets).

GSGDATA (p. 877) Specifies the reference point and defines the geometry in the fiber direction for the generalized plane strain element option.

IMESH (p. 935) Generates nodes and interface elements along lines or areas.

KATT (p. 969) Associates attributes with the selected, unmeshed keypoints.

KCLEAR (p. 975) Deletes nodes and point elements associated with selected keypoints.

KESIZE (p. 978) Specifies the edge lengths of the elements nearest a keypoint.

KMESH (p. 985) Generates nodes and point elements at keypoints.

KREFINE (p. 990) Refines the mesh around specified keypoints.

KSCON (p. 994) Specifies a keypoint about which an area mesh will be skewed.

LATT (p. 1014) Associates element attributes with the selected, unmeshed lines.

LCCAT (p. 1020) Concatenates multiple lines into one line for mapped meshing.

LCLEAR (p. 1024) Deletes nodes and line elements associated with selected lines.

LESIZE (p. 1041) Specifies the divisions and spacing ratio on unmeshed lines.

LMESH (p. 1057) Generates nodes and line elements along lines.

LREFINE (p. 1062) Refines the mesh around specified lines.

MAT (p. 1111) Sets the element material attribute pointer.

MCHECK (p. 1115) Checks mesh connectivity.

MODMSH (p. 1138) Controls the relationship of the solid model and the FE model.

MOPT (p. 1151) Specifies meshing options.

MSHAPE (p. 1188) For elements that support multiple shapes, specifies the element shape to be used for meshing.

MSHCOPY (p. 1189) Simplifies the generation of meshes that have matching node element patterns on two different line groups (in 2D) or area groups (3D).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands mesh solid models with nodes and elements.

MSHKEY (p. 1190) Specifies whether free meshing or mapped meshing should be used to mesh a model.

MSHMID (p. 1191) Specifies placement of midside nodes.

MSHPATTERN (p. 1192) Specifies pattern to be used for mapped triangle meshing.

NREFINE (p. 1257) Refines the mesh around specified nodes.

PSMESH (p. 1543) Splits an initially continuous group of beam, shell, plane, or solid elements into two unconnected groups, tied together with PRETS179 pretension elements.

REAL (p. 1579) Sets the element real constant set attribute pointer.

RSMESH (p. 1621) Generates a result section.

SHPP (p. 1787) Controls element shape checking.

SMRTSIZE (p. 1811) Specifies meshing parameters for automatic (smart) element sizing.

TCHG (p. 1994) Converts 20-node degenerate tetrahedral elements to their 10-node non-degenerate counterparts.

TIMP (p. 2003) Improves the quality of tetrahedral elements that are not associated with a volume.

TYPE (p. 2036) Sets the element type attribute pointer.

VATT (p. 2075) Associates element attributes with the selected, unmeshed volumes.

VCLEAR (p. 2076) Deletes nodes and volume elements associated with selected volumes.

VIMP (p. 2132) Improves the quality of the tetrahedral elements in the selected volume(s).

VMESH (p. 2140) Generates nodes and volume elements within volumes.

VEORIENT (p. 2089) Specifies brick element orientation for volume mapped (hexahedron) meshing.

VSWEEP (p. 2168) Fills an existing unmeshed volume with elements by sweeping the mesh from an adjacent area throughout the volume.

Table 2.38: Nodes

These PREP7 commands create, modify, list, etc., nodes.

CENTER (p. 309) Defines a node at the center of curvature of 2 or 3 nodes.

FILL (p. 755) Generates a line of nodes between two existing nodes.

MOVE (p. 1158) Calculates and moves a node to an intersection.

N (p. 1207) Defines a node.

NAXIS (p. 1208) Generates or clears nodes for general axisymmetric element sections.

NANG (p. 1208) Rotates a nodal coordinate system by direction cosines.

NDELE (p. 1211) Deletes nodes.

NDIST (p. 1212) Calculates and lists the distance between two nodes.

NGEN (p. 1218) Generates additional nodes from a pattern of nodes.

NKPT (p. 1219) Defines a node at an existing keypoint location.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands create, modify, list, etc., nodes.

NLIST (p. 1239) Lists nodes.
NMODIF (p. 1247) Modifies an existing node.
NORA (p. 1252) Rotates nodal coordinate systems to surface normal.
NPLOT (p. 1254) Displays nodes.
NREAD (p. 1256) Reads nodes from a file.
NROTAT (p. 1264) Rotates nodal coordinate systems into the active system.
NRRANG (p. 1265) Specifies the range of nodes to be read from the node file.
NSCALE (p. 1265) Generates a scaled set of nodes from a pattern of nodes.
NSMOOTH (p. 1280) Smooths selected nodes among selected elements.
NSYM (p. 1289) Generates a reflected set of nodes.
NWRITE (p. 1305) Writes nodes to a file.
QUAD (p. 1562) Generates a quadratic line of nodes from three nodes.
SOURCE (p. 1824) Defines a default location for undefined nodes or keypoints.
TRANSFER (p. 2011) Transfers a pattern of nodes to another coordinate system.

Table 2.39: Elements

These PREP7 commands create, modify, list, etc., elements.

AFSURF (p. 106) Generates surface elements overlaid on the surface of existing solid elements and assigns the extra node as the closes fluid element node.
E (p. 591) Defines an element by node connectivity.
ECPCHG (p. 592) Optimizes degree-of-freedom usage in a coupled acoustic model.
EDELE (p. 593) Deletes selected elements from the model.
EEMBED (p. 595) Generates bonded connections between intersecting elements.
EGEN (p. 603) Generates elements from an existing pattern.
EGID (p. 605) Specifies a global identifier for MESH200 elements in a mesh-independent reinforcing analysis.
EINFIN (p. 606) Generates structural infinite elements from selected nodes.
EINTF (p. 614) Defines two-node elements between coincident nodes.
ELIST (p. 619) Lists the elements.
EMID (p. 624) Adds or removes midside nodes.
EMODIF (p. 626) Modifies a previously defined element.
EMORE (p. 627) Adds more nodes to the just-defined element.
EMSEL (p. 628) Selects a set of reinforcing elements via a global identifier.
EMTGEN (p. 630) Generates a set of TRANS126 elements
EN (p. 633) Defines an element by its number and node connectivity.
ENDRELEASE (p. 637) Specifies degrees of freedom to be decoupled for end release.
ENGEN (p. 642) Generates elements from an existing pattern.
ENORM (p. 643) Reorients shell element normals or line element node connectivity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands create, modify, list, etc., elements.

ENSYM (p. 644) Generates elements by symmetry reflection.

EORIENT (p. 646) Reorients solid element normals.

EPLOT (p. 648) Produces an element display.

EREAD (p. 653) Reads elements from a file.

EREINF (p. 656) Generates reinforcing elements from selected existing (base) elements.

ERRANG (p. 660) Specifies the element range to be read from a file.

ESURF (p. 682) Generates elements overlaid on the free faces of selected nodes.

ESYM (p. 684) Generates elements from a pattern by a symmetry reflection.

ESYS (p. 685) Sets the element coordinate system attribute pointer.

EWRITE (p. 703) Writes elements to a file.

GCDEF (p. 788) Defines interface interactions between general contact surfaces.

GCGEN (p. 792) Creates contact elements for general contact.

INISTATE (p. 938) Applies initial state data to an element or a selection of elements.

LAYLIST (p. 1017) Lists real constants material properties for layered elements.

LAYPLOT (p. 1018) Displays the layer stacking sequence for layered elements.

LFSURF (p. 1045) Generates surface elements overlaid on the edge of existing solid elements and assigns the extra node as the closest fluid element node.

MAT (p. 1111) Sets the element material attribute pointer.

NDSURF (p. 1213) Generates surface elements overlaid on the edge of existing elements and assigns the extra node as the closest fluid element node.

REAL (p. 1579) Sets the element real constant set attribute pointer.

SHSD (p. 1791) Creates or deletes shell-solid interface to be used in shell-to-solid assemblies.

SWADD (p. 1889) Adds more surfaces to an existing spot weld set.

SWDEL (p. 1889) Deletes spot weld sets.

SWGEN (p. 1890) Creates a new spot weld set.

SWLIST (p. 1893) Lists spot weld sets.

TSHAP (p. 2023) Defines simple 2D and 3D geometric surfaces for target segment elements.

TYPE (p. 2036) Sets the element type attribute pointer.

UPGEOM (p. 2058) Adds displacements from a previous analysis and updates the geometry to the deformed configuration.

USRDOF (p. 2064) Specifies the degrees of freedom for the user-defined element USER300.

USRELEM (p. 2065) Specifies the characteristics of the user-defined element USER300.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands create, modify, list, etc., elements.

WTBCREATE (p. 2168) Creates and defines USER300 elements for use in aeroelastic analysis.

Table 2.40: Superelements

These PREP7 commands create and modify superelements.

SE (p. 1651) Defines a superelement.

SEDLIST (p. 1705) Lists the degree-of-freedom solution of a superelement after the use pass.

SELIST (p. 1708) Lists the contents of a superelement matrix file.

SESYMM (p. 1718) Performs a symmetry operation on a superelement within the use pass.

SETRAN (p. 1730) Creates a superelement from an existing superelement.

Table 2.41: Coupled Degrees of Freedom

These PREP7 commands define, modify, list, etc., coupled degrees of freedom.

CP (p. 421) Defines (or modifies) a set of coupled degrees of freedom.

CPDELE (p. 424) Deletes coupled degree of freedom sets.

CPINTF (p. 425) Defines coupled degrees of freedom at an interface.

CPLGEN (p. 427) Generates sets of coupled nodes from an existing set.

CPLIST (p. 427) Lists the coupled degree of freedom sets.

CPMERGE (p. 428) Merges different couple sets with duplicate degrees of freedom into one couple set.

CPNGEN (p. 430) Defines, modifies, or adds to a set of coupled degrees of freedom.

CPSEL (p. 431) Selects coupled degree-of-freedom sets via predefined reference numbers.

CPSGEN (p. 432) Generates sets of coupled nodes from existing sets.

Table 2.42: Constraint Equations

These PREP7 commands define, modify, list, etc., constraint equations.

CE (p. 297) Defines a constraint equation relating degrees of freedom.

CECYCMS (p. 300) Generates the constraint equations for a multistage cyclic symmetry analysis.

CEDELE (p. 304) Deletes constraint equations.

CEFILTER (p. 304) Bypasses the error message when constraint equations have excessive terms.

CEIMS (p. 306) Generates constraint equations at the interface between stages in a multistage cyclic symmetry analysis.

CEINTF (p. 305) Generates constraint equations at an interface.

CELIST (p. 308) Lists the constraint equations.

CESEL (p. 313) Selects constraint equations via predefined reference numbers.

CERIG (p. 310) Defines a rigid region.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands define, modify, list, etc., constraint equations.

CESGEN (p. 314)	Generates a set of constraint equations from existing sets.
RBE3 (p. 1573)	Distributes the force/moment applied at an independent node to a set of dependent nodes.

Table 2.43: Status

These PREP7 commands are for use with the STAT command.

AREAS (p. 178)	Specifies "Areas" as the subsequent status topic.
BOOL (p. 268)	Specifies "Booleans" as the subsequent status topic.
CEQN (p. 310)	Specifies "Constraint equations" as the subsequent status topic.
COUPLE (p. 420)	Specifies "Node coupling" as the subsequent status topic.
ELEM (p. 618)	Specifies "Elements" as the subsequent status topic.
ETYPE (p. 703)	Specifies "Element types" as the subsequent status topic.
FEBODY (p. 745)	Specifies "Body loads on elements" as the subsequent status topic.
FECONS (p. 745)	Specifies "Constraints on nodes" as the subsequent status topic.
FEFOR (p. 746)	Specifies "Forces on nodes" as the subsequent status topic.
FESURF (p. 746)	Specifies "Surface loads on elements" as the subsequent status topic.
GEOMETRY (p. 797)	Specifies "Geometry" as the subsequent status topic.
KEYPTS (p. 980)	Specifies "Keypoints" as the subsequent status topic.
LINE (p. 1051)	Specifies "Lines" as the subsequent status topic.
MATER (p. 1112)	Specifies "Material properties" as the subsequent status topic.
MESHING (p. 1124)	Specifies "Meshing" as the subsequent status topic.
NODES (p. 1248)	Specifies "Nodes" as the subsequent status topic.
PRIM (p. 1495)	Specifies "Solid model primitives" as the subsequent status topic.
RCON (p. 1574)	Specifies "Real constants" as the subsequent status topic.
SELM (p. 1709)	Specifies "Superelements" as the subsequent status topic.
TBLE (p. 1988)	Specifies "Data table properties" as the subsequent status topic.
VOLUMES (p. 2141)	Specifies "Volumes" as the subsequent status topic.

Table 2.44: Sections

These PREP7 commands manage sections.

BSAX (p. 271)	Specifies axial strain and force for a nonlinear general beam section.
BSM1 (p. 273)	Specifies bending moment and curvature in plane XZ for a nonlinear general beam section.
BSM2 (p. 274)	Specifies bending moment and curvature in plane XY for a nonlinear general beam section.
BSMD (p. 272)	Specifies mass density for a nonlinear general beam section.
BSS1 (p. 276)	Specifies transverse shear force and strain in plane XZ for a nonlinear general beam section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands manage sections.

BSS2 (p. 277) Specifies transverse shear force and strain in plane XY for a nonlinear general beam section.

BSTE (p. 278) Specifies a thermal expansion coefficient for a nonlinear general beam section.

BSTQ (p. 278) Specifies cross section torque and twist for a nonlinear general beam section.

CBMD (p. 287) Specifies preintegrated mass-density matrix for composite-beam sections.

CBMX (p. 288) Specifies preintegrated cross-section stiffness for composite-beam sections.

CBTE (p. 289) Specifies a thermal expansion coefficient for a composite-beam section.

CBTMP (p. 290) Specifies a temperature for the composite-beam section matrix.

SDELETE (p. 1650) Deletes sections from the database.

SECCONTROL (p. 1652) Supplements or overrides default section properties.

SECDATA (p. 1655) Describes the geometry of a section.

SECJOINT (p. 1679) Defines local coordinate systems at joint element nodes and other data for joint elements.

/SECLIB (p. 1682) Sets the default section library path for the **SECREAD (p. 1693)** command.

SECLOCK (p. 1682) Specifies locks on the element degrees of freedom.

SECNUM (p. 1685) Sets the element section attribute pointer.

SECOFFSET (p. 1685) Defines the section offset for cross sections.

SECPLOT (p. 1688) Plots the geometry of a beam, pipe, elbow, shell, or reinforcing section to scale.

SECREAD (p. 1693) Reads a custom section library or a user-defined section mesh into Mechanical APDL.

SECSTOP (p. 1696) Specifies stops on the element degrees of freedom.

SECTYPE (p. 1697) Associates section type information with a section ID number.

SECWRITE (p. 1703) Creates an ASCII file containing user mesh section information.

SFLEX (p. 1779) Sets section flexibility factors for pipe elements.

SLIST (p. 1795) Summarizes the section properties for all defined sections in the current session of Mechanical APDL.

SLOAD (p. 1797) Loads a pretension section.

SSBT (p. 1846) Specifies the bending thermal effects quantity for a preintegrated shell section.

SSMT (p. 1849) Specifies the membrane thermal effects quantity for a preintegrated shell section.

SSPA (p. 1851) Specifies the membrane stiffness quantity for a preintegrated shell section.

SSPB (p. 1852) Specifies the coupling stiffness quantity for a preintegrated shell section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands manage sections.

SSPD (p. 1853) Specifies the bending stiffness quantity for a preintegrated shell section.

SSPE (p. 1854) Specifies the transverse shear stiffness quantity for a preintegrated shell section.

SSPM (p. 1855) Specifies the mass density and stress-free initial temperature for a preintegrated shell section.

Table 2.45: Morphing

These PREP7 commands adjust the finite element mesh in the non-structural regions to coincide with the deflections of the structural regions.

MORPH (p. 1156) Specifies morphing and remeshing controls.

DAMORPH (p. 494) Move nodes in selected areas to conform to structural displacements.

DEMORPH (p. 516) Move nodes in selected elements to conform to structural displacements.

DVMORPH (p. 588) Move nodes in selected volumes to conform to structural displacements.

Table 2.46: Artificially Matched Layers

These PREP7 commands create artificially matched layers (PMLs or AMLs) designed to absorb high frequency waves.

PMLOPT (p. 1451) Defines perfectly matched layers (PMLs) or irregular perfectly matched layers (IPMLs).

PMLSIZE (p. 1453) Determines the number of PML or IPML layers.

PSYS (p. 1554) Sets the PML element coordinate system attribute pointer.

Table 2.47: Special Purpose

These PREP7 commands are used for special-purpose operations.

AEROCOEFF (p. 102) Computes the aero-damping and stiffness coefficients and writes them to an APDL array

CINT (p. 334) Defines parameters associated with fracture parameter calculations.

/CYCEXPAND (p. 454) Verifies a cyclically symmetric model by graphically expanding it partially or into the full 360 degrees.

CYCFREQ (p. 458) Specifies solution options for a cyclic symmetry mode superposition harmonic analysis.

CYCLIC (p. 462) Specifies a cyclic symmetry analysis.

CYCOPT (p. 466) Specifies solution options for a cyclic symmetry analysis.

EMSYM (p. 629) Specifies circular symmetry for electromagnetic sources.

MSOPT (p. 1195) Specifies solution options for a multistage cyclic symmetry analysis.

MSTOLE (p. 1200) Adds two extra nodes from FLUID116 elements to SURF152 elements for convection analyses.

PERBC2D (p. 1376) Generates periodic constraints for 2D planar magnetic field analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These PREP7 commands are used for special-purpose operations.

RACE (p. 1566)	Defines a "racetrack" current source.
SSTATE (p. 1855)	Defines a steady-state rolling analysis.
XFCRKMESH (p. 2191)	Defines a crack in the model when the crack surface is discretized by MESH200 elements.
XFDATA (p. 2192)	Defines a crack in the model by specifying nodal level set values.
XFENRICH (p. 2193)	Defines parameters associated with crack propagation using XFEM.
XFLIST (p. 2194)	Lists enrichment details and associated crack information.

2.6. SOLUTION Commands

These commands load and solve the model. The commands are grouped by functionality.

Table 2.48: Analysis Options

These SOLUTION commands set general analysis options.

ABEXTRACT (p. 88)	Extracts the alpha-beta damping multipliers for Rayleigh damping.
ACCOPTION (p. 91)	Specifies GPU accelerator capability options.
ADAMS (p. 94)	Performs solutions and writes flexible body information to a modal neutral file.
ADPCI (p. 98)	Defines parameters associated with adaptive crack initiation.
ANTYPE (p. 162)	Specifies the analysis type and restart status.
ASCRES (p. 192)	Specifies the output type for an acoustic scattering analysis.
ASOL (p. 200)	Activate a specified acoustic solution.
BCSOPTION (p. 218)	Sets memory option for the sparse solver.
CECHECK (p. 299)	Check constraint equations and couplings for rigid body motions.
CEFILTER (p. 304)	Bypasses the error message when constraint equations have excessive terms.
CHECK (p. 333)	Checks current database items for completeness.
CINT (p. 334)	Defines parameters associated with contour integral calculations.
CJUMP (p. 347)	Initiates a cycle-jump analysis.
CLOAD (p. 351)	Initiates a cyclic-loading analysis.
CMATRIX (p. 360)	Performs electrostatic field solutions and calculates the self and mutual capacitances between multiple conductors.
CMSOPT (p. 378)	Specifies component mode synthesis (CMS) analysis options.
CNCHECK (p. 382)	Provides and/or adjusts the initial status of contact pairs.
CNKMOD (p. 392)	Modifies contact element key options.
CNTR (p. 395)	Redirects contact pair output quantities to a text file.
CUTCONTROL (p. 425)	Controls time-step cutback during a nonlinear solution.
CYCOPT (p. 466)	Specifies solution options for a cyclic symmetry analysis.
DAMPOPT (p. 495)	Sets damped eigensolver options.
DDOPTION (p. 508)	Sets domain decomposer option for DMP solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands set general analysis options.

DMPEXT (p. 557) Extracts modal damping coefficients in a specified frequency range.

DMPOPTION (p. 559) Specifies distributed-memory parallel (DMP) file combination options.

DSPOPTION (p. 576) Sets the memory option for the distributed sparse solver.

EXBOPT (p. 705) Specifies `.exb` file output options in a CMS generation pass.

EMATWRITE (p. 623) Forces the writing of all the element matrices to `File.emat`.

EQSLV (p. 648) Specifies the type of equation solver.

ERESX (p. 658) Specifies extrapolation of integration point results.

ESCHECK (p. 660) Perform element shape checking for a selected element set.

EXPASS (p. 714) Specifies an expansion pass of an analysis.

GAUGE (p. 787) Gauges the problem domain for an edge-element formulation.

GMATRIX (p. 862) Performs electric field solutions and calculates the self and mutual conductances between multiple conductors.

INVOPT (p. 957) Enables or disables inverse solving for the current load step.

KRYOPT (p. 992) Specifies solution options for a Krylov method harmonic analysis.

LANBOPTION (p. 1007) Specifies Block Lanczos eigensolver options.

LUMPM (p. 1093) Specifies a lumped mass matrix formulation.

MODDIR (p. 1134) Activates the remote read-only modal files usage.

MONITOR (p. 1145) Controls contents of variable fields in the nonlinear solution monitor file.

MSAVE (p. 1183) Sets the memory saving feature for the PCG solver.

MSOLVE (p. 1193) Starts multiple solutions for an acoustic analysis.

OPNCONTROL (p. 1226) Sets decision parameter for automatically increasing the time step interval in a pure thermal analysis.

OUTAERO (p. 1329) Outputs the superelement matrices and load vectors to formatted files for aeroelastic analysis.

PCGOPT (p. 1365) Controls PCG solver options.

PERTURB (p. 1378) Sets linear perturbation analysis options.

PRSCONTROL (p. 1520) Specifies whether to include pressure load stiffness in the element stiffness formation.

PSCONTROL (p. 1526) Turns off shared-memory parallel operations during solution.

RATE (p. 1571) Specifies whether the effect of creep strain rate will be used in the solution of a load step.

RESVEC (p. 1603) Calculates residual vectors in a modal analysis, or includes residual vectors in a modal superposition transient/harmonic analysis.

RSTOFF (p. 1630) Offsets node or element IDs in the FE geometry record.

SCOPT (p. 1649) Specifies System Coupling options.

SEEXP (p. 1705) Specifies options for the substructure expansion pass.

SEOPT (p. 1715) Specifies substructure analysis options.

SNOPTION (p. 1815) Specifies supernode (SNODE) eigensolver options.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands set general analysis options.

/SOLU (p. 1821) Enters the solution processor.

SOLVE (p. 1822) Starts a solution.

STABILIZE (p. 1858) Activates stabilization for all elements that support nonlinear stabilization.

THEXPAND (p. 1996) Enables or disables thermal loading.

THOPT (p. 1997) Nonlinear transient thermal solution option.

TOFFST (p. 2008) Specifies the temperature offset from absolute zero to zero.

Table 2.49: Nonlinear Options

These SOLUTION commands define options for nonlinear analyses.

ARCLEN (p. 175) Activates the arc-length method.

ARCTRM (p. 177) Controls termination of the solution when the arc-length method (**ARCLEN (p. 175)**,ON) is used.

BUCOPT (p. 280) Specifies buckling analysis options.

CNVTOL (p. 396) Sets convergence values for nonlinear analyses.

/GST (p. 880) Turns Graphical Solution Tracking (GST) on or off.

LNSRCH (p. 1057) Activates a line search to be used with Newton-Raphson.

MXPAND (p. 1203) Specifies modal or buckling analysis expansion options.

NCNV (p. 1210) Sets the key to terminate an analysis.

NEQIT (p. 1214) Specifies the maximum number of equilibrium iterations for nonlinear analyses.

NLADAPTIVE (p. 1216) Defines the criteria under which the mesh is refined or modified during a nonlinear solution.

NLDIAG (p. 1224) Sets nonlinear diagnostics functionality.

NLGEOM (p. 1231) Includes large deformation effects in a static or full transient analysis.

NLHIST (p. 1232) Specify results items to track during solution.

NLMESH (p. 1240) Controls remeshing in nonlinear adaptivity.

NROPT (p. 1261) Specifies the Newton-Raphson options in a static or full transient analysis.

PRED (p. 1476) Activates a predictor in a nonlinear analysis.

PSTRES (p. 1551) Specifies whether prestress effects are calculated or included.

SEMIIMPLICIT (p. 1710) Specifies parameters for a semi-implicit solution.

SOLOPTION (p. 1857) **Specifies solution transition options.**

Table 2.50: Dynamic Options

These SOLUTION commands define options for dynamic analyses.

AEROCOEFF (p. 102) Computes the aero-damping and stiffness coefficients and writes them to an APDL array

ALPHAD (p. 115) Defines the mass matrix multiplier for damping.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define options for dynamic analyses.

BETAD (p. 220) Defines the stiffness matrix multiplier for damping.

CECYCMS (p. 300) Generates the constraint equations for a multistage cyclic symmetry analysis.

CYCFREQ (p. 458) Specifies solution options for a cyclic symmetry mode-superposition harmonic analysis.

CEIMS (p. 306) Generates constraint equations at the interface between stages in a multistage cyclic symmetry analysis.

DMPRAT (p. 562) Sets a constant modal damping ratio.

DMPSTR (p. 563) Sets a constant structural damping coefficient.

FRQSCL (p. 777) Turns on automatic scaling of the entire mass matrix and frequency range for modal analyses.

HARFRQ (p. 885) Defines the frequency range in the harmonic analysis.

HREXP (p. 911) Specifies the phase angle for the harmonic analysis expansion pass.

HROCEAN (p. 917) Includes ocean wave effects in a harmonic analysis.

HROPT (p. 912) Specifies harmonic analysis options.

HROUT (p. 918) Specifies the harmonic analysis output options.

LVSCALE (p. 1094) Scales the load vector for mode-superposition analyses.

MASCALE (p. 1110) Turns on the scaling of the entire system matrix.

MCFOPT (p. 1116) Specifies options for the Modal Coordinates File (*Jobname.mcf*).

MDAMP (p. 1117) Defines the damping ratios as a function of mode.

MDPLOT (p. 1119) Plots frequency-dependent modal damping coefficients.

MIDTOL (p. 1127) Sets middle step residual criterion values for structural transient analysis.

MODCONT (p. 1132) Creates multiple load vectors in a modal analysis.

MODOPT (p. 1140) Specifies modal analysis options.

MSOPT (p. 1195) Specifies solution options for a multistage cyclic symmetry analysis.

MXPAND (p. 1203) Specifies modal or buckling analysis expansion options.

QRDOPT (p. 1560) Specifies additional QRDAMP modal analysis option.

RIGID (p. 1609) Specifies known rigid body modes (if any) of the model.

SUBOPT (p. 1869) Specifies SUBSPACE eigensolver options

TIMINT (p. 2001) Turns on transient effects.

TINTP (p. 2004) Defines transient integration parameters.

TRNOPT (p. 2017) Specifies transient analysis options.

Table 2.51: Spectrum Options

These SOLUTION commands define options for spectrum analyses.

ADDAM (p. 96) Specifies the acceleration spectrum computation constants for the analysis of shock resistance of shipboard structures.

COVAL (p. 420) Defines PSD cospectral values.

CQC (p. 433) Specifies the complete quadratic mode combination method.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define options for spectrum analyses.

DSUM (p. 578) Specifies the double sum mode combination method.

FREQ (p. 776) Defines the frequency points for the SV vs. FREQ tables.

GRP (p. 873) Specifies the grouping mode combination method.

MMASS (p. 1131) Specifies the missing mass response calculation.

NRLSUM (p. 1258) Specifies the Naval Research Laboratory (NRL) sum mode combination method.

PFACT (p. 1385) Calculates participation factors for the PSD or multi-point response spectrum table.

PSDCOM (p. 1527) Specifies the power spectral density mode combination method.

PSDFRQ (p. 1528) Defines the frequency points for the input spectrum vs. FREQ tables of PSD and multi-point spectrum analyses.

PSDGRAPH (p. 1529) Displays input PSD curves

PSDRES (p. 1530) Controls solution output written to the results file from a PSD analysis.

PSDSPL (p. 1532) Defines a partially correlated excitation in a PSD analysis.

PSDUNIT (p. 1532) Defines the type of PSD or multi-point response spectrum.

PSDVAL (p. 1534) Defines PSD or multi-point response spectrum values.

PSDWAV (p. 1534) Defines a wave propagation excitation in a PSD analysis.

QDVAL (p. 1559) Defines PSD quadspectral values.

ROCK (p. 1614) Specifies a rocking response spectrum.

ROSE (p. 1615) Specifies the Rosenblueth mode combination method.

RIGRESP (p. 1610) Selects the spectrum type and other spectrum options.

SED (p. 1704) Defines the excitation direction for a single-point response spectrum.

SPDAMP (p. 1827) Defines input spectrum damping in a multi-point response spectrum analysis.

SPFREQ (p. 1828) Defines the frequency points for the input spectrum tables **SPVAL (p. 1842)** vs. **SPFREQ (p. 1828)** for multi-point spectrum analyses.

SPGRAPH (p. 1829) Displays input spectrum curves for MPRS analysis.

SPOPT (p. 1837) Selects the spectrum type and other spectrum options.

SPUNIT (p. 1841) Defines the type of multi-point response spectrum.

SPVAL (p. 1842) Defines multi-point response spectrum values.

SRSS (p. 1845) Specifies the square root of sum of squares mode combination method.

SV (p. 1886) Defines spectrum values to be associated with frequency points.

SVPLOT (p. 1886) Displays input spectrum curves.

SVTYP (p. 1887) Defines the type of single-point response spectrum.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define options for spectrum analyses.

VDDAM (p. 2081) Specifies the velocity spectrum computation constants for the analysis of shock resistance of shipboard structures.

Table 2.52: Load Step Options

These SOLUTION commands define options for individual load steps.

AUTOTS (p. 207) Specifies whether to use automatic time stepping or load stepping.

CAMPBELL (p. 284) Prepares the result file for a subsequent Campbell diagram of a prestressed structure.

CECMOD (p. 300) Modifies the constant term of a constraint equation during solution.

DELTIM (p. 515) Specifies the time step sizes to be used for this load step.

EXPSOL (p. 721) Specifies the solution to be expanded for reduced analyses.

KBC (p. 970) Specifies stepped or ramped loading within a load step.

KUSE (p. 1002) Specifies whether or not to reuse the factorized matrix.

MAGOPT (p. 1102) Specifies options for a 3D magnetostatic field analysis.

MAGSOLV (p. 1103) Specifies magnetic solution options and initiates the solution.

MODE (p. 1136) Specifies the harmonic loading term for this load step.

NSUBST (p. 1287) Specifies the number of substeps to be taken this load step.

NUMEXP (p. 1293) Specifies solutions to be expanded from reduced analyses.

TIME (p. 2000) Sets the time for a load step.

TREF (p. 2013) Defines the reference temperature for the thermal strain calculations.

TSRES (p. 2026) Defines an array of key times at which the time-stepping strategy changes.

UPCOORD (p. 2056) Modifies the coordinates of the active set of nodes, based on the current displacements.

USRCAL (p. 2062) Enables user-solution subroutines to be activated or deactivated.

WRFULL (p. 2184) Stops solution after assembling global matrices.

Table 2.53: Solid Constraints

These SOLUTION commands define constraints on the solid model.

DA (p. 487) Defines symmetry or antisymmetry degree-of-freedom constraints on areas.

DADELE (p. 491) Deletes degree-of-freedom constraints on an area.

DALIST (p. 493) Lists the degree-of-freedom constraints on an area.

DK (p. 541) Defines degree-of-freedom constraints at keypoints.

DKDELE (p. 543) Deletes degree-of-freedom constraints at a keypoint.

DKLIST (p. 545) Lists the degree-of-freedom constraints at keypoints.

DL (p. 545) Defines symmetry or antisymmetry degree-of-freedom constraints on lines.

DLDELE (p. 548) Deletes degree-of-freedom constraints on a line.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define constraints on the solid model.

DLLIST (p. 551)	Lists degree-of-freedom constraints on a line.
DTRAN (p. 583)	Transfers solid model degree-of-freedom constraints to the finite element model.

Table 2.54: Solid Forces

These SOLUTION commands define forces on the solid model.

FK (p. 762)	Defines force loads at keypoints.
FKDELE (p. 764)	Deletes force loads at a keypoint.
FKLIST (p. 765)	Lists the forces at keypoints.
FTRAN (p. 783)	Transfers solid model forces to the finite element model.

Table 2.55: Solid Surface Loads

These SOLUTION commands define surface loads on the solid model.

SFA (p. 1739)	Specifies surface loads on the selected areas.
SFADELE (p. 1744)	Deletes surface loads from areas.
SFALIST (p. 1745)	Lists the surface loads for the specified area.
SFL (p. 1774)	Specifies surface loads on lines of an area.
SFLDELE (p. 1777)	Deletes surface loads from lines.
SFLLIST (p. 1781)	Lists the surface loads for lines.
SFTRAN (p. 1782)	Transfer the solid model surface loads to the finite element model.

Table 2.56: Solid Body Loads

These SOLUTION commands define body loads on the solid model.

BFA (p. 229)	Defines a body-force load on an area.
BFADELE (p. 231)	Deletes body-force loads on an area.
BFALIST (p. 233)	Lists the body-force loads on an area.
BFK (p. 248)	Defines a body-force load at a keypoint.
BFKDELE (p. 250)	Deletes body-force loads at a keypoint.
BFKLIST (p. 251)	Lists the body-force loads at keypoints.
BFL (p. 252)	Defines a body-force load on a line.
BFLDELE (p. 254)	Deletes body-force loads on a line.
BFLLIST (p. 256)	Lists the body-force loads on a line.
BFTRAN (p. 259)	Transfers solid model body-force loads to the finite element model.
BFV (p. 261)	Defines a body-force load on a volume.
BFVDELE (p. 263)	Deletes body-force loads on a volume.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define body loads on the solid model.

BFVLIST (p. 264) Lists the body-force loads on a volume.

Table 2.57: Inertia

These SOLUTION commands define inertial loads on the model.

ACEL (p. 92) Specifies the linear acceleration of the structure.

AIRL (p. 111) Specifies that automatic inertia relief calculations are to be performed.

CGLOC (p. 319) Specifies the origin location of the acceleration coordinate system.

CGOMGA (p. 320) Specifies the rotational velocity of the global origin.

CMACEL (p. 358) Specifies the translational acceleration of an element component.

CMDOMEGA (p. 362) Specifies the rotational acceleration of an element component about a user-defined rotational axis.

CMOMEGA (p. 368) Specifies the rotational velocity of an element component about a user-defined rotational axis.

CMROTATE (p. 373) Specifies the rotational velocity of an element component about a user-defined rotational axis

CORIOLIS (p. 417) Applies the Coriolis effect to a rotating structure.

DCGOMG (p. 501) Specifies the rotational acceleration of the global origin.

DOMEGA (p. 571) Specifies the rotational acceleration of the structure.

IRLF (p. 960) Specifies that inertia relief calculations are to be performed.

OMEGA (p. 1324) Specifies the rotational velocity of the structure.

SPOINT (p. 1836) Defines a point for moment summations.

SYNCHRO (p. 1893) Specifies whether the excitation frequency is synchronous or asynchronous with the rotational velocity of the structure.

Table 2.58: Miscellaneous Loads

These SOLUTION commands are for miscellaneous load definition and control.

ANPRES (p. 148) Produces an animated sequence of the time-harmonic pressure variation of an engine-order excitation in a cyclic harmonic analysis.

APORT (p. 170) Specifies input data for plane wave and acoustic duct ports.

ASIFILE (p. 195) Writes or reads one-way acoustic-structural coupling data.

AWAVE (p. 213) Specifies input data for an acoustic incident wave.

BIOT (p. 265) Calculates the Biot-Savart source magnetic field intensity.

DFSWAVE (p. 529) Specifies the incident planar waves with random phases for the diffuse sound field.

FLUREAD (p. 769) Reads one-way Fluent-to-Mechanical APDL coupling data via a `.cgns` file with one-side fast Fourier transformation complex pressure peak value.

IC (p. 921) Specifies initial conditions at nodes.

ICDELE (p. 923) Deletes initial conditions at nodes.

ICLIST (p. 923) Lists the initial conditions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands are for miscellaneous load definition and control.

ICROTATE (p. 925) Specifies initial velocity at nodes as a sum of rotation about an axis and translation.

INISTATE (p. 938) Applies initial state data to an element or a selection of elements.

MPCHG (p. 1167) Changes the material number attribute of an element.

MRPM (p. 1183) Defines the revolutions per minute (RPM) for a machine rotation.

NSMASS (p. 1277) Applies nonstructural mass to selected elements.

OSRESULT (p. 1327) Controls the selected result data written to the database.

OUTGEOM (p. 1330) Controls geometry-related data written to the results file.

OUTPR (p. 1332) Controls the solution printout.

OUTRES (p. 1336) Controls the solution data written to the database.

RESCONTROL (p. 1592) Controls file writing for multiframe restarts.

RSTCONTROL (p. 1624) Controls whether element single value results are written to the results file.

SBCLIST (p. 1646) Lists solid model boundary conditions.

SBCTRAN (p. 1647) Transfers solid model loads and boundary conditions to the FE model.

WSPRINGS (p. 2187) Creates weak springs on corner nodes of a bounding box of the currently selected elements.

XFLIST (p. 2194) Lists enrichment details and associated crack information.

Table 2.59: Load Step Operations

These SOLUTION commands write and solve multiple load steps.

LSCLEAR (p. 1071) Clears loads and load step options from the database.

LSDELE (p. 1073) Deletes load step files.

LSREAD (p. 1083) Reads load and load step option data into the database.

LSSOLVE (p. 1085) Reads and solves multiple load steps.

LSWRITE (p. 1087) Writes load and load step option data to a file.

Table 2.60: Master Degrees of Freedom

These SOLUTION commands define master degrees of freedom.

M (p. 1097) Defines master degrees of freedom for reduced and superelement generation analyses.

MDELE (p. 1118) Deletes master degrees of freedom.

MGEN (p. 1127) Generates additional master degrees of freedom from a previously defined set.

MLIST (p. 1130) Lists the master degrees of freedom.

Table 2.61: Rezoning

These SOLUTION commands apply to analyses that use rezoning.

REZONE (p. 1606) Initiates a rezoning operation, sets rezoning options, and rebuilds the database.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands apply to analyses that use rezoning.

MAPSOLVE (p. 1108)	Maps the solved node and element solutions from an original mesh to a new mesh.
MAPVAR (p. 1109)	Defines tensors and vectors in user-defined state variables (for user-defined materials or user-defined creep laws).
REMESH (p. 1581)	Specifies the starting and ending remeshing points.
AREMESH (p. 180)	Generates an area for creating a new mesh.

Table 2.62: 2D to 3D Analysis

These SOLUTION commands apply to 2D to 3D analyses.

MAP2DTO3D (p. 1107)	Initiates the 2D to 3D analysis process.

Table 2.63: Birth and Death

These SOLUTION commands are used for the birth and death option for elements.

EALIVE (p. 592)	Reactivates an element (for the birth and death capability).
EKILL (p. 616)	Deactivates an element (for the birth and death capability).
ESTIF (p. 681)	Specifies the matrix multiplier for deactivated elements.

Table 2.64: FE Constraints

These SOLUTION commands define constraints on the finite element model.

D (p. 483)	Defines degree-of-freedom constraints at nodes.
DCUM (p. 502)	Specifies that degree-of-freedom constraint values are to be accumulated.
DDELE (p. 505)	Deletes degree-of-freedom constraints.
DFLX (p. 528)	Imposes a uniform magnetic flux B on an edge-element electromagnetic model.
DJ (p. 537)	Specify boundary conditions on the components of relative motion of a joint element.
DJDELE (p. 539)	Deletes boundary conditions on the components of relative motion of a joint element.
DJLIST (p. 541)	Lists boundary conditions applied to joint elements.
DLIST (p. 550)	Lists degree-of-freedom constraints.
DSCALE (p. 574)	Scales degree-of-freedom constraint values.
DSYM (p. 580)	Specifies symmetry or antisymmetry degree-of-freedom constraints on nodes.
DVAL (p. 587)	Defines values at enforced group.
GSBDATA (p. 876)	Specifies the constraints or applies the load at the ending point for the generalized plane strain element option.
GSLIST (p. 878)	When using generalized plane strain, lists the input data or solutions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define constraints on the finite element model.

LDREAD (p. 1036) Reads results from the results file and applies them as loads.

Table 2.65: FE Forces

These SOLUTION commands define nodal loads on the finite element model.

F (p. 729) Specifies force loads at nodes.

FCUM (p. 739) Specifies that force loads are to be accumulated.

FDELE (p. 742) Deletes force loads on nodes.

FJ (p. 759) Specify forces or moments on the components of the relative motion of a joint element.

FJDELE (p. 760) Deletes forces (or moments) on the components of the relative motion of a joint element.

FJLIST (p. 762) Lists forces and moments applied on joint elements.

FLIST (p. 766) Lists force loads on the nodes.

FSCALE (p. 778) Scales force load values in the database.

Table 2.66: FE Surface Loads

These SOLUTION commands define surface loads on the finite element model.

SF (p. 1733) Defines surface loads on nodes.

SFBEAM (p. 1746) Specifies surface loads on beam elements.

SFCONTROL (p. 1758) Defines surface load properties on selected elements and nodes for subsequent **SF (p. 1733)** or **SFE (p. 1761)** loading commands.

SFCUM (p. 1758) Specifies that surface loads are to be accumulated.

SFDELE (p. 1760) Deletes surface loads.

SFE (p. 1761) Defines surface loads on elements.

SFEDELE (p. 1768) Deletes surface loads from elements.

SFELIST (p. 1770) Lists the surface loads for elements.

SFFUN (p. 1771) Specifies a varying surface load.

SFGRAD (p. 1772) Specifies a gradient (slope) for surface loads.

SFLIST (p. 1780) Lists surface loads.

SFSCALE (p. 1781) Scales surface loads on elements.

Table 2.67: FE Body Loads

These SOLUTION commands define body loads on the finite element model.

BF (p. 221) Defines a nodal body-force load.

BFCUM (p. 233) Specifies that nodal body-force loads are to be accumulated.

BFDELE (p. 235) Deletes nodal body-force loads.

BFE (p. 237) Defines an element body-force load.

BFECUM (p. 241) Specifies whether to ignore subsequent element body-force loads.

BFEDELE (p. 243) Deletes element body-force loads.

BFELIST (p. 244) Lists the element body-force loads.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define body loads on the finite element model.

BFESCAL (p. 245) Scales element body-force loads.

BFLIST (p. 255) Lists the body-force loads on nodes.

BFPORT (p. 257) Transfers a thermal body-force load (HGEN) load from selected MESH200 elements to reinforcing elements.

BFSCALE (p. 257) Scales body-force loads at nodes.

BFUNIF (p. 259) Assigns a uniform body-force load to all nodes.

LDREAD (p. 1036) Reads results from the results file and applies them as loads.

TUNIF (p. 2027) Assigns a uniform temperature to all nodes.

Table 2.68: Ocean

These SOLUTION commands define ocean load data.

OCDATA (p. 1307) Defines an ocean load using non-table data.

OCDELETE (p. 1314) Deletes a previously defined ocean load.

OCLIST (p. 1315) Summarizes all currently defined ocean loads.

OCREAD (p. 1315) Reads externally defined ocean data.

OCTABLE (p. 1316) Defines an ocean load using table data.

OCTYPE (p. 1320) Specifies the type of ocean load data to follow (basic, current, or wave).

OCZONE (p. 1322) Specifies the type of ocean zone data to follow (component, Z-level, or pipe)

Table 2.69: Status

These SOLUTION commands are for use with the STAT (p. 1861) command.

ATYPE (p. 206) Specifies "Analysis types" as the subsequent status topic.

BIOOPT (p. 264) Specifies "Biot-Savart options" as the subsequent status topic.

DEACT (p. 510) Specifies "Element birth and death" as the subsequent status topic.

DYNOPT (p. 590) Specifies "Dynamic analysis options" as the subsequent status topic.

GENOPT (p. 796) Specifies "General options" as the subsequent status topic.

INRTIA (p. 955) Specifies "Inertial loads" as the subsequent status topic.

LSOPER (p. 1080) Specifies "Load step operations" as the subsequent status topic.

MASTER (p. 1111) Specifies "Master DOF" as the subsequent status topic.

NLOPT (p. 1247) Specifies "Nonlinear analysis options" as the subsequent status topic.

OUTOPT (p. 1332) Specifies "Output options" as the subsequent status topic.

SMBODY (p. 1806) Specifies "Body loads on the solid model" as the subsequent status topic.

SMCONS (p. 1807) Specifies "Constraints on the solid model" as the subsequent status topic.

SMFOR (p. 1807) Specifies "Forces on the solid model" as the subsequent status topic.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands are for use with the STAT (p. 1861) command.

SMSURF (p. 1814) Specifies "Surface loads on the solid model" as the subsequent status topic.

SOLUOPT (p. 1821) Specifies "Solution options" as the subsequent status topic.

SPTOPT (p. 1840) Specifies "Spectrum analysis options" as the subsequent status topic.

Table 2.70: Radiosity

These SOLUTION commands calculate the radiation view factors and to specify the solution parameters for the Radiosity solver method.

HEMIOPT (p. 904) Specifies options for Hemicube view factor calculation.

RADOPT (p. 1567) Specifies Radiosity Solver options.

RDEC (p. 1575) Defines the decimation parameters used by the radiosity solver method.

RSOPT (p. 1622) Creates or loads the radiosity mapping data file for SURF251 or SURF252 element types.

RSURF (p. 1631) Generates the radiosity surface elements and stores them in the database.

RSYMM (p. 1632) Defines symmetry, rotation, or extrusion parameters for the radiosity method.

SPCNOD (p. 1825) Defines a space node for radiation using the Radiosity method.

SPCTEMP (p. 1826) Defines a free-space ambient temperature for radiation using the Radiosity method.

STEF (p. 1866) Specifies Stefan-Boltzmann radiation constant.

V2DOPT (p. 2070) Specifies 2D/axisymmetric view factor calculation options.

VFDI (p. 2095) Specifies diagnostic options to check that the row sum relationship is satisfied.

VFOPT (p. 2099) Specifies options for view factor file.

QSOPT (p. 1561) Specifies quasi static radiation options.

VFUP (p. 2110) Controls view factor updating at the substep level for a coupled-field radiation analysis that includes large-deflection effects.

Table 2.71: Additive Manufacturing

These SOLUTION commands define options for additive manufacturing analysis.

AMBEAM (p. 117) For multiple-beam printers, specifies the number of beams.

AMBUILD (p. 118) Specifies printer parameters for the build and other options.

AMENV (p. 120) Specifies the build-environment thermal boundary conditions.

AMMAT (p. 121) Specifies the melting and relaxation temperatures of the build material in an additive manufacturing analysis.

AMPOWDER (p. 122) Specifies the powder thermal conditions.

AMRESULT (p. 123) Specifies result data written to an `AMResults.txt` file.

AMSTEP (p. 124) Specifies the process-sequence steps.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These SOLUTION commands define options for additive manufacturing analysis.

AMSUPPORTS (p. 127)	Specifies the support information.
AMTYPE (p. 128)	Specifies the printing process.

2.7. POST1 Commands

These commands postprocess the results with the database processor. The commands are grouped by functionality.

Table 2.72: Setup

These POST1 commands put data into the database for postprocessing.

APPEND (p. 173)	Reads data from the results file and appends it to the database.
DESOL (p. 520)	Defines or modifies solution results at a node of an element.
DNSOL (p. 564)	Defines or modifies solution results at a node.
FILE (p. 752)	Specifies the data file where results are to be found.
HRCPLX (p. 910)	Computes and stores in the database the time-harmonic solution at a prescribed phase angle.
/POST1 (p. 1461)	Enters the database results postprocessor.
RESCOMBINE (p. 1589)	Reads results from local results files into the database after a distributed-memory parallel (DMP) solution.
RESET (p. 1598)	Resets all POST1 or POST26 specifications to initial defaults.
SET (p. 1724)	Defines the data set to be read from the results file.
SUBSET (p. 1871)	Reads results for the selected portions of the model.

Table 2.73: Controls

These POST1 commands control the calculations done for other commands.

AVPRIN (p. 210)	Specifies how principal and vector sums are to be calculated.
AVRES (p. 212)	Specifies how results data will be averaged when PowerGraphics is enabled.
/EFACET (p. 602)	Specifies the number of facets per element edge for PowerGraphics displays.
ERNORM (p. 659)	Controls error estimation calculations.
FORCE (p. 772)	Selects the element nodal force type for output.
INRES (p. 953)	Identifies the data to be retrieved from the results file.
LAYER (p. 1015)	Specifies the element layer for which data are to be processed.
RSYS (p. 1639)	Activates a coordinate system for printout or display of results.
SHELL (p. 1784)	Selects a shell element or shell layer location for results output.

Table 2.74: Results

These POST1 commands process results, such as degree-of-freedom results, nodal stresses, and element summable and nonsummable miscellaneous data.

NSORT (p. 1283)	Sorts nodal data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST1 commands process results, such as degree-of-freedom results, nodal stresses, and element summable and nonsummable miscellaneous data.

NUSORT (p. 1303) Restores original order for nodal data.

PLCINT (p. 1396) Plots the contour integral (**CINT (p. 334)**) result data.

PLCKSURF (p. 1399) Plots the $\Phi = 0$ level set surface in an XFEM-based crack analysis

PLDISP (p. 1400) Displays the displaced structure.

PLESOL (p. 1401) Displays solution results as discontinuous element contours.

PLNSOL (p. 1425) Displays solution results as continuous element contours.

PLORB (p. 1437) Displays the orbital motion of a rotating structure undergoing vibration.

PRENERGY (p. 1478) Prints the total energies of a model.

PRORB (p. 1515) Prints the orbital motion characteristics of a rotating structure undergoing vibration.

PLVECT (p. 1445) Displays results as vectors.

PRCINT (p. 1472) Lists the contour integral (**CINT (p. 334)**) result data.

PRESOL (p. 1481) Prints the solution results for elements.

PRJSOL (p. 1498) Print joint element output.

PRNLD (p. 1503) Prints the summed element nodal loads.

PRNSOL (p. 1505) Prints the nodal solution results.

PRRFOR (p. 1517) Used with the **FORCE (p. 772)** command. Prints the constrained node reaction solution.

PRRSOL (p. 1518) Prints the constrained node reaction solution.

PRVECT (p. 1524) Prints results as vector magnitude and direction cosines.

SUMTYPE (p. 1880) Sets the type of summation to be used in the following load case operations.

Table 2.75: Element Table

These POST1 commands operate with the element table, which in turn is used to process results stored for each element, such as average stresses, heat fluxes, etc.

DETAB (p. 522) Modifies element table results in the database.

ESORT (p. 680) Sorts the element table.

ETABLE (p. 687) Fills a table of element values for further processing.

EUSORT (p. 703) Restores original order of the element table.

PLETAB (p. 1411) Displays element table items.

PLLS (p. 1419) Displays element table items as contoured areas along elements.

PLVECT (p. 1445) Displays results as vectors.

PRETAB (p. 1490) Prints the element table items.

PRVECT (p. 1524) Prints results as vector magnitude and direction cosines.

SABS (p. 1643) Specifies absolute values for element table operations.

SADD (p. 1643) Forms an element table item by adding two existing items.

SALLOW (p. 1644) Defines the allowable stress table for safety factor calculations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST1 commands operate with the element table, which in turn is used to process results stored for each element, such as average stresses, heat fluxes, etc.

SEXP (p. 1732) Forms an element table item by exponentiating and multiplying.

SFACT (p. 1743) Enables safety factor or margin of safety calculations to be made.

SFCALC (p. 1747) Calculates the safety factor or margin of safety.

SMAX (p. 1805) Forms an element table item from the maximum of two other items.

SMIN (p. 1808) Forms an element table item from the minimum of two other items.

SMULT (p. 1814) Forms an element table item by multiplying two other items.

SSUM (p. 1858) Calculates and prints the sum of element table items.

TALLOW (p. 1897) Defines the temperature table for safety factor calculations.

VCROSS (p. 2079) Forms element table items from the cross product of two vectors.

VDOT (p. 2083) Forms an element table item from the dot product of two vectors.

Table 2.76: Listing

These POST1 commands control printed listings of results.

/FORMAT (p. 774) Specifies format controls for tables.

/HEADER (p. 902) Sets page and table heading print controls.

IRLIST (p. 962) Prints inertia relief summary table.

/PAGE (p. 1348) Defines the printout and screen page size.

PRERR (p. 1480) Prints SEPC and TEPC.

PRITER (p. 1497) Prints solution summary data.

Table 2.77: Animation

These POST1 commands animate results.

ANCNTR (p. 131) Produces an animated sequence of a contoured deformed shape.

ANCUT (p. 132) Produces an animated sequence of Q-slices.

ANDATA (p. 134) Produces a sequential contour animation over a range of results data.

ANDSCL (p. 136) Produces an animated sequence of a deformed shape.

ANCYC (p. 133) Applies a traveling wave animation to graphics data in a modal cyclic symmetry analysis.

ANDYNA (p. 137) Produces an animated sequence of contour values through substeps.

/ANFILE (p. 138) Saves or resumes an animation sequence to or from a file.

ANFLOW (p. 139) Produces an animated sequence of a charged particle traveling in an electric or magnetic field.

ANHARM (p. 141) Produces an animated sequence of time-harmonic results or complex mode shapes.

ANIM (p. 142) Displays graphics data in animated form.

ANISOS (p. 143) Produces an animated sequence of an isosurface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST1 commands animate results.

ANMODE (p. 144) Produces an animated sequence of a mode shape.

ANTIME (p. 161) Generates a sequential contour animation over a range of time.

TRTIME (p. 2022) Defines the options used for the **PLTRAC (p. 1442)** (charged particle trace) command.

Table 2.78: Path Operations

These POST1 commands are used for path operations.

PADELE (p. 1347) Deletes a defined path.

PAGET (p. 1349) Writes current path information into an array variable.

PAPUT (p. 1350) Retrieves path information from an array variable.

PARESU (p. 1350) Restores previously saved paths from a file.

PASAVE (p. 1353) Saves selected paths to an external file.

PATH (p. 1354) Defines a path name and establishes parameters for the path.

PCALC (p. 1362) Forms additional labeled path items by operating on existing path items.

PCROSS (p. 1371) Calculates the cross product of two path vectors along the current path.

PDEF (p. 1372) Interpolates an item onto a path.

PDOT (p. 1375) Calculates the dot product of two path vectors along the current path.

PLPAGM (p. 1438) Displays path items along the path geometry.

PLPATH (p. 1439) Displays path items on a graph.

PLSECT (p. 1440) Displays membrane and membrane-plus-bending linearized stresses.

PMAP (p. 1449) Creates mapping of the path geometry by defining path interpolation division points.

PPATH (p. 1463) Defines a path by picking or defining nodes, or locations on the currently active working plane, or by entering specific coordinate locations.

PRANGE (p. 1464) Determines the path range.

PRPATH (p. 1517) Prints path items along a geometry path.

PRSECT (p. 1520) Calculates and prints linearized stresses along a section path.

PSEL (p. 1536) Selects a path or paths.

PVECT (p. 1556) Interpolates a set of items onto a path.

Table 2.79: Surface Operations

These POST1 commands define an arbitrary surface and to develop results information for that surface.

SUCALC (p. 1873) Create new result data by operating on two existing result datasets on a given surface.

SUCR (p. 1875) Create a surface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST1 commands define an arbitrary surface and to develop results information for that surface.

SUDEL (p. 1877)	Delete geometry information as well as any mapped results for specified surface or for all selected surfaces.
SUEVAL (p. 1877)	Perform operations on a mapped item and store result in a scalar parameter.
SUGET (p. 1878)	Create and dimension an NPT row array parameter named PARM, where NPT is the number of geometry points in SurfName.
SUMAP (p. 1879)	Map results onto selected surface(s).
SUPL (p. 1881)	Plot specified SetName result data on all selected surfaces or on the specified surface.
SUPR (p. 1881)	Print surface information.
SURESU (p. 1882)	Resume surface definitions from a specified file.
SUSAVE (p. 1883)	Save surface definitions and result items to a file.
SUSEL (p. 1884)	Selects a subset of surfaces
SUVECT (p. 1885)	Operate between two mapped result vectors.

Table 2.80: Load Case Calculations

These POST1 commands are used for combining results from different load steps.

LCABS (p. 1018)	Specifies absolute values for load case operations.
LCASE (p. 1019)	Reads a load case into the database.
LCDEF (p. 1021)	Creates a load case from a set of results on a results file.
LCFACT (p. 1022)	Defines scale factors for load case operations.
LCFILE (p. 1023)	Creates a load case from an existing load case file.
LCOPER (p. 1026)	Performs load case operations.
LCSEL (p. 1029)	Selects a subset of load cases.
LCSUM (p. 1031)	Specifies whether to process nonsummable items in load case operations.
LCWRITE (p. 1031)	Creates a load case by writing results to a load case file.
LCZERO (p. 1032)	Zeroes the results portion of the database.
RAPPND (p. 1570)	Appends results data from the database to the results file.

Table 2.81: Magnetics Calculations

These POST1 commands are used for special purpose magnetics postprocessing.

CURR2D (p. 444)	Calculates current flow in a 2D conductor.
EMAGERR (p. 622)	Calculates the relative error in an electrostatic or electromagnetic field analysis.
EMF (p. 623)	Calculates the electromotive force (emf), or voltage drop along a predefined path.
EMFT (p. 624)	Summarizes electromagnetic forces and torques on a selected set of nodes.
FLUXV (p. 768)	Calculates the flux passing through a closed contour.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST1 commands are used for special purpose magnetics postprocessing.

MMF (p. 1132) Calculates the magnetomotive force along a path.

PLF2D (p. 1417) Generates a contour line plot of equipotentials.

POWERH (p. 1462) Calculates the rms power loss in a conductor or lossy dielectric.

SENERGY (p. 1714) Determines the stored magnetic energy or co-energy.

Table 2.82: Trace Points

These POST1 commands trace charged particle motion.

PLTRAC (p. 1442) Displays a charged particle trace on an element display.

TRPDEL (p. 2020) Deletes charged particle trace points.

TRPLIS (p. 2020) Lists charged particle trace points.

TRPOIN (p. 2021) Defines a point through which a charged particle trace will travel.

Table 2.83: Special Purpose

These POST1 commands are used for various special purposes.

BFINT (p. 246) Activates the body force interpolation operation.

CBDOF (p. 285) Activates cut boundary interpolation (for submodeling).

CMSFILE (p. 376) Specifies the component mode synthesis (CMS) results files to include when plotting the mode shape of an assembly.

CYCCALC (p. 453) Calculates results from a cyclic harmonic mode-superposition analysis using the specifications defined by **CYCSPEC (p. 475)**.

/CYCEXPAND (p. 454) Graphically expands displacements, stresses and strains of a cyclically symmetric model partially or through the full 360 degrees.

CYCFILES (p. 457) Specifies the data files where results are to be found for a cyclic symmetry mode-superposition harmonic analysis.

CYCPHASE (p. 472) Provides tools for determining minimum and maximum possible result values from frequency couplets produced in a modal cyclic symmetry analysis.

CYCSPEC (p. 475) Defines the set of result items for a subsequent **CYCCALC (p. 453)** command in postprocessing a cyclic harmonic mode-superposition analysis.

EXOPTION (p. 709) Specifies the **EXPROFILE (p. 719)** options for the Mechanical APDL to Ansys CFX profile file transfer.

/EXPAND (p. 712) Enables the creation of a larger graphic display than that which is represented by the actual finite element analysis model.

EXPAND (p. 711) Displays the results of a modal cyclic symmetry analysis.

EXPROFILE (p. 719) Exports interface loads to a CFX Profile file.

EXUNIT (p. 725) Indicates units assumed for an interface load for Mechanical APDL to CFX transfer.

FSSPARM (p. 780) Calculates reflection and transmission properties of a frequency selective surface.

FSUM (p. 780) Sums the nodal force and moment contributions of elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST1 commands are used for various special purposes.

HFANG (p. 904)	Defines or displays spatial angles of a spherical radiation surface for antenna parameter calculations.
HFSYM (p. 905)	Sets symmetry planes for the computation of high-frequency acoustic fields in the near and far field domains (beyond the finite element region).
MACOPT (p. 1098)	Specifies modal assurance criterion (MAC) calculation options for **RSTMAC (p. 1627)**.
MAXCYCMODE (p. 1112)	Computes the maximum results value of cyclic mode pairs at the same frequency or the amplitude of a single complex mode.
MSOPT (p. 1195)	Specifies solution options for a multistage cyclic symmetry analysis
NFORCE (p. 1216)	Sums the nodal forces and moments of elements attached to nodes.
NLDPOST (p. 1229)	Gets element component information from nonlinear diagnostic files.
PLAS (p. 1387)	Plots a specified acoustic quantity during postprocessing of an acoustic analysis.
PLCAMP (p. 1392)	Plots a Campbell diagram for applications involving rotating structure dynamics.
PLCFREQ (p. 1395)	Plots the frequency response for the given **CYCSPEC (p. 475)** specification
PLCHIST (p. 1395)	Plots a histogram of the frequency response of each sector for the given **CYCSPEC (p. 475)** specification.
PLFAR (p. 1412)	Plots electric far fields and far field parameters.
PLMC (p. 1421)	Plots the modal coordinates from a mode-superposition solution.
PLNEAR (p. 1423)	Plots the electric field in the near zone exterior to the equivalent source surface.
PLZZ (p. 1448)	Plots the interference diagram from a cyclic modal analysis.
PRAS (p. 1464)	Prints a specified acoustic quantity during postprocessing of an acoustic analysis.
PRCAMP (p. 1469)	Prints a Campbell diagram for applications involving rotating structure dynamics.
PRFAR (p. 1490)	Prints electric far fields and far field parameters.
PRMC (p. 1499)	Prints the modal coordinates from a mode-superposition solution.
PRNEAR (p. 1501)	Prints the electric field in the near zone exterior to the equivalent source surface.
RESWRITE (p. 1604)	Appends results data from the database to a results file in a top-down substructure analysis.
RMFLVEC (p. 1612)	Writes eigenvectors of fluid nodes to a file for use in damping parameter extraction.
RSPLIT (p. 1623)	Creates one or more results file(s) from the current results file based on subsets of elements.
SPOINT (p. 1836)	Defines a point for moment summations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST1 commands are used for various special purposes.

SPMWRITE (p. 1833)	Calculates state-space matrices and writes them to the `.spm` file.

Table 2.84: Status

These POST1 commands are for use with the STAT (p. 1861) command.

CALC (p. 283)	Specifies "Calculation settings" as the subsequent status topic.
DATADEF (p. 498)	Specifies "Directly defined data status" as the subsequent status topic.
DEFINE (p. 511)	Specifies "Data definition settings" as the subsequent status topic.
DISPLAY (p. 535)	Specifies "Display settings" as the subsequent status topic.
LCCALC (p. 1020)	Specifies "Load case settings" as the subsequent status topic.
POINT (p. 1459)	Specifies "Point flow tracing settings" as the subsequent status topic.
PRINT (p. 1496)	Specifies "Print settings" as the subsequent status topic.
SORT (p. 1824)	Specifies "Sort settings" as the subsequent status topic.
SPEC (p. 1828)	Specifies "Miscellaneous specifications" as the subsequent status topic.

Table 2.85: Failure Criteria

These POST1 commands are for use with the failure criteria.

FC (p. 732)	Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.
FCCHECK (p. 735)	Checks both the strain and stress input criteria for all materials.
FCDELE (p. 736)	Deletes previously defined failure criterion data for the given material.
FCLIST (p. 737)	Lists the failure criteria that has been input.
FCTYP (p. 740)	Activates or removes failure-criteria types for postprocessing.

2.8. POST26 Commands

These commands postprocess the results with the time-history processor. The commands are grouped by functionality.

Table 2.86: Setup

These POST26 commands store data for processing.

ANSOL (p. 149)	Specifies averaged element nodal data to be stored from the results file.
CISOL (p. 345)	Stores J-integral information in a variable.
DATA (p. 497)	Reads data records from a file into a variable.
ENERSOL (p. 641)	Specifies the total energies to be stored.
ESOL (p. 672)	Specifies element data to be stored from the results file.
FILE (p. 752)	Specifies the data file where results are to be found.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST26 commands store data for processing.

GSSOL (p. 879)	Specifies which results to store from the results file when using generalized plane strain.
JSOL (p. 966)	Specifies result items to be stored for the joint element.
NSOL (p. 1281)	Specifies nodal data to be stored from the results file.
NSTORE (p. 1287)	Defines which time points are to be stored.
NUMVAR (p. 1302)	Specifies the number of variables allowed in POST26.
/POST26 (p. 1462)	Enters the time-history results postprocessor.
RESET (p. 1598)	Resets all POST1 or POST26 specifications to initial defaults.
RFORCE (p. 1607)	Specifies the total reaction force data to be stored.
/RGB (p. 1608)	Specifies the RGB color values for indices and contours.
SOLU (p. 1819)	Specifies solution summary data per substep to be stored.
STORE (p. 1867)	Stores data in the database for the defined variables.
TIMERANGE (p. 2001)	Specifies the time range for which data are to be stored.
VARDEL (p. 2074)	Deletes a variable (GUI).
VARNAM (p. 2075)	Names (or renames) a variable.

Table 2.87: Controls

These POST26 commands control the calculations of other commands.

CFACT (p. 315)	Defines complex scaling factors to be used with operations.
FORCE (p. 772)	Selects the element nodal force type for output.
LAYERP26 (p. 1016)	Specifies the element layer for which data are to be stored.
SHELL (p. 1784)	Selects a shell element or shell layer location for results output.
TVAR (p. 2029)	Changes time to the cumulative iteration number.

Table 2.88: Operations

These POST26 commands perform operations on the stored variables.

ABS (p. 89)	Forms the absolute value of a variable.
ADD (p. 95)	Adds variables.
ATAN (p. 204)	Forms the arctangent of a complex variable.
CLOG (p. 354)	Forms the common log of a variable
CONJUG (p. 414)	Forms the complex conjugate of a variable.
DERIV (p. 518)	Differentiates a variable.
EXP (p. 710)	Forms the exponential of a variable.
FILLDATA (p. 756)	Fills a variable by a ramp function.
IMAGIN (p. 934)	Forms an imaginary variable from a complex variable.
INT1 (p. 956)	Integrates a variable.
LARGE (p. 1013)	Finds the largest (the envelope) of three variables.
NLOG (p. 1246)	Forms the natural log of a variable.
PROD (p. 1515)	Multiplies variables.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST26 commands perform operations on the stored variables.

QUOT (p. 1563) Divides two variables.

REALVAR (p. 1579) Forms a variable using only the real part of a complex variable.

SMALL (p. 1800) Finds the smallest of three variables.

SQRT (p. 1843) Forms the square root of a variable.

Table 2.89: Display

These POST26 commands display the results.

KEEP (p. 977) Stores POST26 definitions and data during active session.

PLCPLX (p. 1399) Specifies the part of a complex variable to display.

PLTIME (p. 1441) Defines the time range for which data are to be displayed.

PLVAR (p. 1444) Displays up to ten variables in the form of a graph.

SPREAD (p. 1840) Turns on a dashed tolerance curve for the subsequent curve plots.

XVAR (p. 2199) Specifies the X variable to be displayed.

Certain graphics commands (p. 16) also apply, such as **/AXLAB (p. 216)**, **/XRANGE (p. 2199)**, and **/YRANGE (p. 2201)**.

Table 2.90: Listing

These POST26 commands produce tabular listings of the results.

EXTREM (p. 725) Lists the extreme values for variables.

LINES (p. 1052) Specifies the length of a printed page.

NPRINT (p. 1255) Defines which time points stored are to be listed.

PRCPLX (p. 1476) Defines the output form for complex variables.

PRTIME (p. 1522) Defines the time range for which data are to be listed.

PRVAR (p. 1523) Lists variables vs. time (or frequency).

Table 2.91: Special Purpose

These POST26 commands are used for various special purposes.

CVAR (p. 451) Computes covariance between two quantities.

PMGTRAN (p. 1450) Summarizes electromagnetic results from a transient analysis.

RCYC (p. 1574) Calculates cyclic results for a mode-superposition harmonic solution.

RESP (p. 1599) Generates a response spectrum.

RPSD (p. 1619) Computes response power spectral density (PSD).

SMOOTH (p. 1809) Enables smoothing of noisy data and provides a graphical representation of the data.

VGET (p. 2129) Moves a variable into an array parameter vector.

VPUT (p. 2152) Moves an array parameter vector into a variable.

Table 2.92: Status

These POST26 commands are for use with the STAT (p. 1861) command.

DEFINE (p. 511) Specifies "Data definition settings" as the subsequent status topic.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These POST26 commands are for use with the STAT (p. 1861) command.	
OPERATE (p. 1325)	Specifies "Operation data" as the subsequent status topic.
PLOTTING (p. 1438)	Specifies "Plotting settings" as the subsequent status topic.
PRINT (p. 1496)	Specifies "Print settings" as the subsequent status topic.

2.9. AUX2 Commands

These commands examine or manipulate the contents of binary files produced by the program. The commands are grouped by functionality.

Table 2.93: Binary File Dump

These AUX2 commands dump the contents of binary files.	
/AUX2 (p. 208)	Enters the binary file dumping processor.
DUMP (p. 583)	Dumps the contents of a binary file.
FILEAUX2 (p. 753)	Specifies the binary file to be dumped.
FORM (p. 773)	Specifies the format of the file dump.
PTR (p. 1555)	Dumps the record of a binary file.

Table 2.94: Binary File Manipulation

These AUX2 commands manipulate the contents of binary files.	
COMBINE (p. 406)	Combines distributed-memory parallel (DMP) files.
HBMAT (p. 888)	Writes an assembled global matrix in Harwell-Boeing format.
PSMAT (p. 1540)	Writes an assembled global matrix to a postscript format that graphically displays nonzero matrix values.

2.10. AUX3 Commands

The auxiliary processor /AUX3 (p. 209) allows you to operate on results files by deleting sets or by changing values. AUX3 commands affect only the data steps index (DSI), time (TIM), loadstep, substep and cumulative step iteration (LSP) records in the results file.

Table 2.95: Results Files

These commands are related to the /AUX3 (p. 209) command.	
/AUX3 (p. 209)	Enters the results file editing processor.
COMPRESS (p. 408)	Deletes all specified sets.
DELETE (p. 513)	Specifies sets in the results file to be deleted before postprocessing.
FILEAUX3 (p. 754)	Specifies the results file to be edited.
LIST (p. 1054)	Lists out the sets in the results file.
MODIFY (p. 1137)	Changes the listed values of the data in a set.
UNDELETE (p. 2052)	Removes results sets from the group of sets selected for editing.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.11. AUX12 Commands

These commands define radiation options for use in thermal analyses. The commands are grouped by functionality.

Table 2.96: General Radiation

These AUX12 commands specify general radiation parameters.	
/AUX12 (p. 209)	Enters the radiation matrix generation processor.
STEF (p. 1866)	Specifies Stefan-Boltzmann radiation constant.
TOFFST (p. 2008)	Specifies the temperature offset from absolute zero to zero.

Table 2.97: Radiation Matrix Method

These AUX12 commands calculate radiation view factors and to create a radiation substructure for the Radiation Matrix method.	
EMIS (p. 625)	Specifies the emissivity as a material property.
GEOM (p. 796)	Defines the geometry specifications for the radiation matrix calculation.
MPRINT (p. 1178)	Specifies that radiation matrices are to be printed.
SPACE (p. 1825)	Defines a space node for radiation.
VTYPE (p. 2172)	Specifies the viewing procedure used to determine the form factors.
WRITE (p. 2185)	Writes the radiation matrix file.

Table 2.98: Radiosity Solver

These AUX12 commands calculate the radiation view factors and to specify the solution parameters for the Radiosity solver method.	
HEMIOPT (p. 904)	Specifies options for Hemicube view factor calculation.
RADOPT (p. 1567)	Specifies Radiosity Solver options.
SPCNOD (p. 1825)	Defines a space node for radiation using the Radiosity method.
SPCTEMP (p. 1826)	Defines a free-space ambient temperature for radiation using the Radiosity method.
V2DOPT (p. 2070)	Specifies 2D/axisymmetric view factor calculation options.
VFDI (p. 2095)	Specifies diagnostic options to check that the row sum relationship is satisfied.
VFOPT (p. 2099)	Specifies options for view factor file.
VFQUERY (p. 2102)	Queries and prints element Hemicube view factors and average view factor.
VFSM (p. 2104)	Adjusts view factor matrix to satisfy reciprocity and/or row sum properties (view factor smoothing).
VFUP (p. 2110)	Controls view factor updating at the substep level for a coupled-field radiation analysis that includes large-deflection effects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.12. AUX15 Commands

These commands read in an IGES file for analysis in Mechanical APDL. The commands are grouped by functionality.

Table 2.99: IGES

These AUX15 commands read an IGES format file into PREP7 data.	
/AUX15 (p. 210)	Enters the IGES file transfer processor.
IGESIN (p. 931)	Transfers IGES data from a file into Mechanical APDL.
IOPTN (p. 957)	Controls options relating to importing a model.

2.13. Mapping Processor Commands

The **/MAP (p. 1107)** processor allows you to map data from an external file onto the existing geometry.

Table 2.100: Pressure Mapping

These commands read pressure data from an external file and map it onto a geometry.	
FTYPE (p. 783)	Specifies the file type and the pressure type for the subsequent import of source points and pressures.
MAP (p. 1105)	Maps pressures from source points to target surface elements.
/MAP (p. 1107)	Enters the mapping processor.
PLGEOM (p. 1418)	Plots target and source geometries.
PLMAP (p. 1420)	Plots target and source pressures.
READ (p. 1578)	Reads coordinate and pressure data from a file.
TARGET (p. 1897)	Specifies the target nodes for mapping pressure onto surface effect elements.
WRITEMAP (p. 2186)	Writes interpolated pressure data to a file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Command Dictionary

This chapter contains a dictionary of the Mechanical APDL commands, listed in alphabetical order. The star (*) and slash (/) of the star and slash commands are ignored for alphabetization (for example, the **/SHOW (p. 1785)** command appears between the **SHELL (p. 1784)** and **SHPP (p. 1787)** commands). As in a dictionary, keywords are located at the top of each page (in the printed version only) indicating the first and last commands contained on that page.

The following section documents the components of a command description.

Components of a Command Description

SAMPLECMD, *Key*

Specifies whether to use automatic time stepping or load stepping.

SOLUTION: Load Step Options

Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Key

Automatic time stepping key:

OFF --

Do not use automatic time stepping

ON --

Use automatic time stepping.

Default: No automatic time stepping.

Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used. Used only if *DTIME* (specified on the **DELTIM (p. 515)** command) is less than the time span or conversely, if *NSBSTP* (on the **NSUBST (p. 1287)** command) is greater than one.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Product Restrictions

Ansys Mechanical Enterprise

Key = OFF is only valid with the Ansys Mechanical Enterprise family of products (Ansys Mechanical Enterprise, Ansys Mechanical Enterprise PrepPost, and Ansys Mechanical Enterprise Solver).

Menu Paths

Main Menu> Preprocessor >Loads> Load Step Opts> Time/Frequenc> Freq and Substps

Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time - Time Step

Main Menu> Solution> Load Step Opts> Time/Frequenc> Freq and Substps

Main Menu> Solution> Load Step Opts> Time/Frequenc> Time - Time Step

Features Documented in Mechanical APDL Commands

The first line of the command description shows the command name followed by the argument names (if any). The second line summarizes the command function. The summary is *not* intended to be all inclusive:

SAMPLECMD, *Key*

Specifies whether to use automatic time stepping or load stepping.

Listed on the next line are codes that will help you find other commands in the program with related functionality.

SOLUTION: Load Step Options

Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

There is at least one group code for each command. "SOLUTION: Load Step Options" is the code in the example above. The portion of the code before the colon (:) tells which of the tables in Command Groupings (p. 11) the command can be found in. These tables represent major groupings, such as PREP7 or APDL commands. In the example above, SOLUTION commands are found in SOLUTION Commands (p. 38). Most of these major groupings are processors, and in those cases the command can be entered only when in that processor. Any exceptions are noted under "Notes" later in the description.

The portion of the code after the colon is the subtable in Command Groupings (p. 11) containing the command. The subtables list commands that functionally relate to each other. In the example, "Load Step Options" means that the command can be found in Table 2.52: Load Step Options (p. 43), a subtable of SOLUTION Commands (p. 38), with other load step option commands.

If you are viewing this reference in the Mechanical APDL Help System, just click on the code to follow the link to the corresponding table. Then click on any of the commands in the table, to follow the link to its description.

If you are reading the printed version, simply turn to the appropriate table in Command Groupings (p. 11). The table lists the page number where the command is documented.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The next line displays a series of product codes, which may contain all of the Mechanical APDL product codes:

Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

or a subset of them:

Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

For more information on product codes, see Valid Products (p. 2).

Following this "product code" line is the description of all arguments, if any, of the command:

Key

Automatic time stepping key:

OFF --

Do not use automatic time stepping (default).

ON --

Use automatic time stepping.

The argument is described and, where necessary, valid choices for the argument are described. Many arguments list defaults, which are the values assumed for *that* argument if you *enter the command but leave the argument blank*. On the other hand, shown after the argument descriptions is often another default:

Command Default

No automatic time stepping.

This is the *command default*. This is the specification assumed by the program if you *do not enter the command at all*. Only commands that set specifications (*specification* commands) have defaults listed. Commands that cause some action, such as performing some calculation, are called *action* commands and simply do not perform the action if the command is not entered. Defaults are not listed for action commands.

Following the default listing are any notes about the command. These notes expand on the summary description given up near the command format, and describe any other behavior, restrictions, suggestions, etc. of that command:

Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used. Used only if *DTIME* (specified on the **DELTIM (p. 515)** command) is less than the time span or conversely, if *NSBSTP* (on the **NSUBST (p. 1287)** command) is greater than one.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the command behaves differently, or has restrictions, in any of the various Ansys products, those differences are described in the *Product Restrictions* section:

Product Restrictions

Ansys Mechanical Enterprise

Key = OFF is only valid with the Ansys Mechanical Enterprise family of products (Ansys Mechanical Enterprise, Ansys Mechanical Enterprise PrepPost, and Ansys Mechanical Enterprise Solver).

Menu Paths

Main Menu >Preprocessor> Loads> Load Step Opts> Time/Frequenc> Freq and Substps

Main Menu >Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time - Time Step

Main Menu> Solution> Load Step Opts> Time/Frequenc >Freq and Substps

Main Menu> Solution> Load Step Opts> Time/Frequenc> Time - Time Step

These menu paths are intended as guides to help you find the commands in the GUI. Be aware, however, that often the paths are valid only if some other command has been previously issued. For example, you won't be able to follow a path to transient analysis options if you have not selected a transient analysis in the first place.

Not all commands are directly accessible through the menu (although some of them may be generated indirectly by some menu function). Such cases are indicated by the following statement in the menu path listing:

Menu Paths

This command cannot be accessed directly in the menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Connection Commands

~CAT5IN,*Name*,*Extension*,*Path*,*Entity*,*FMT*,*NOCL*,*NOAN*

Transfers a .CATPart file into the Mechanical APDL program.

CAD Import

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

The name of a valid .CATPart file, created with CATIA Version 5.0. The first character of the file name must be an alphanumeric.

Extension

The extension for the file. The default extension is .CATPart.

Path

The path name of the directory in which the file resides enclosed in single quotes. The default path name is the current working directory.

Entity

Entity to be imported.

SOLIDS

Solids only, imported as Mechanical APDL volumes (default).

SURFACES

Surfaces only, imported as Mechanical APDL areas.

ALL

All entities. Use this option when the file contains different types of entities.

FMT

The format in which Mechanical APDL will store the model.

0

Neutral format (default). Defeaturing after import is restricted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Solid format; this allows defeaturing after import.

NOCL

Remove tiny objects.

0

Remove tiny objects without checking model validity (default).

1

Do not remove tiny objects.

NOAN

Perform an analysis of the model.

0

Analyze the model (default).

1

Do not analyze the model.

Notes

If defeaturing is specified (*FMT* = 1), this command must be the last line of any file, script, or other interactive input.

More information on importing CATIA Version 5 parts is available in CATIA V5 in the *Connection User's Guide*.

Menu Paths

File>Import>CATIA V5

~CATIAIN,*Name*,*Extension*,*Path*,- -,- -,*BLANK*,- -

Transfers a CATIA model into the Mechanical APDL program.

CAD Import

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

The name of a valid CATIA model, created with CATIA 4.x or lower. The first character of the file name must be an alphanumeric. Special characters such as & - and * and spaces are not permitted in the part name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Extension

The extension for the file. The default extension is `.model`.

Path

The path name of the directory in which the file resides, enclosed in single quotes. The default path name is the current working directory.

--

Unused field.

--

Unused field.

BLANK

Sets whether to import "blanked" entities.

0

Does not import "blanked" (suppressed) CATIA entities (default).

1

Imports "blanked" entities. The portions of CATIA data that were suppressed will be included in the import.

--

Unused field.

Notes

More information on importing CATIA parts is available in CATIA V4 in the *Connection User's Guide*.

Menu Paths

File>Import>CATIA

~PARAIN,*Name*,*Extension*,*Path*,*Entity*,*FMT*,*Scale*

Transfers a Parasolid file into Mechanical APDL.

CAD Import

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

The name of a valid Parasolid file. The first character of the file name must be an alphanumeric.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Extension

The extension for the file. The default extension is `.x_t` on a PC or `.xmt_txt` on a Linux system. Parasolid files are compatible across systems, and do not need to be renamed to be used on another platform.

Path

The path name of the directory in which the file resides, enclosed in single quotes. The default path name is the current working directory.

Entity

Entity to be imported:

SOLIDS

Solids only, imported as Mechanical APDL volumes (default)

SURFACES

Surfaces only, imported as Mechanical APDL areas.

WIREFRAME

Wireframe only, imported as Mechanical APDL lines.

ALL

All entities. Use this option when the file contains more than one type of entity.

FMT

Sets the format in which Mechanical APDL will store the model

0

Neutral format (default). Defeaturing after import is restricted. Use this option if you need to scale a model to a specific unit of measure (other than meters).

1

Solid format; this allows defeaturing after import.

Scale

Allows scaling for the model

0

Do not rescale the model; retain the default Parasolid setting of meters (default).

1

Scale the model if warranted by the model size.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Very small models will be scaled by the factor of 10 or 100 to increase the chance of successful import; the scaling factor used is displayed in the output window and in the `.para_log` file. Because scaling changes the dimensions of the model, you must apply loads and material properties appropriately. If the model cannot be properly scaled, the analysis may fail.

If you need to scale your model to a specific set of measurements, set FMT = 0, then use either the **VLSCALE (p. 2137)**, **ARSCALE (p. 183)** or **LSSCALE (p. 1084)** command to select a different unit of measure.

Notes

More information on importing Parasolid parts is available in Parasolid in the *Connection User's Guide*.

Menu Paths

File>Import>PARA

~PROEIN,*Name*,*Extension*,*Path*,*Proecomm*

Transfers a Creo Parametric part into Mechanical APDL.

CAD Import

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

The name of the Creo Parametric part to be imported, which cannot exceed 64 characters in length and must begin with an alphanumeric character. Special characters such as & - and * and spaces are not permitted in the part name.

Extension

The general Creo Parametric extension format is `prt` for parts and `asm` for assemblies.

Path

Full path name to the directory containing the part. The default is the current working directory.

Proecomm

The start command for the version of Creo Parametric you are using. `proe1` is the default command. Note that the full path name to the Creo Parametric command need not be used here if the path had been included in the PATH variable. The Creo Parametric command name is set by the **PROE_START_CMD241** environment variable.

Notes

More information on importing Creo Parametric parts is available in Creo Parametric in the *Connection User's Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

File>Import>Creo Parametric

~SATIN,*Name*,*Extension*,*Path*,*Entity*,*FMT*,*NOCL*,*NOAN*

Transfers a `.sat` file into Mechanical APDL.

CAD Import

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

The name of a valid `.sat` file, created with a supported version of ACIS. The first character of the file name must be an alphanumeric. Special characters such as & - and * and spaces are not permitted in the part name. See File Names (p. 7) in the *Command Reference* (p. 1) for more information about Mechanical APDL file-naming conventions.

Extension

The extension for the file. The default extension is `.sat`.

Path

The path name of the directory in which the file resides enclosed in single quotes. The default path name is the current working directory.

Entity

Entity to be imported.

SOLIDS

Solids only, imported as Mechanical APDL volumes (Not implemented, imports **All**).

SURFACES

Surfaces only, imported as Mechanical APDL areas (Not implemented, imports **All**).

WIREFRAME

Wireframe only, imported as Mechanical APDL lines (Not implemented, imports **All**).

ALL

All entities. Use this option when the file contains different types of entities.

FMT

The format in which Mechanical APDL will store the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Neutral format (default). Defeaturing after import is restricted.

1

Solid format; this allows defeaturing after import.

NOCL

Remove tiny objects.

0

Remove tiny objects without checking model validity (default).

1

Do not remove tiny objects.

NOAN

Perform an ACIS analysis of the model.

0

Analyze the model (default).

1

Do not analyze the model.

Note:

NOCL and *NOAN* are not supported in the Mechanical APDL GUI.

Notes

More information on importing ACIS parts is available in ACIS in the *Connection User's Guide*.

Menu Paths

File>Import>ACIS

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

~UGIN,*Name*,*Extension*,*Path*,*Entity*,*LAYER*,*FMT*

Transfers an NX part into Mechanical APDL.

CAD Import

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

The file name of the NX part to be imported, which cannot exceed 64 characters in length. The path name must begin with an alphanumeric character. Special characters such as &, -, and * are not permitted in the part name.

Extension

The NX part file extension. The default is `.prt`.

Path

The full path name to the directory containing the part, enclosed in single quotes; for example, `'/ug_parts'`. The default is the current working directory.

Entity

Entity to be imported.

0 or Solid

Solids only, imported as Mechanical APDL volumes (the default).

1 or Surface

Surfaces only, imported as Mechanical APDL areas.

2 or Wireframe

Wireframe only, imported as Mechanical APDL lines.

3 or All

All entities. Use this option when the part contains entities that may not be attached to each other, such as a solid in one location and a surface in another.

LAYER

The number(s) assigned to the layer(s) to be imported. You can import one layer or a range of layers (designated by hyphens). Defaults to 1-256 (all layers).

FMT

The format in which Mechanical APDL stores the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Neutral format (default). Defeaturing after import is restricted.

1

Solid format; this allows defeaturing after import.

Notes

More information on importing NX parts is available in UG/NX in the *Connection User's Guide*.

Menu Paths

File>Import>NX

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Plugin Commands

*DPF,*Action*,*WorkflowId*,*Val1*,*Val2*,*Val3*

Specifies an input data processing framework (DPF) workflow file or a DPF workflow output pin.

DPF commands

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Description

This command is added to the list of Mechanical APDL commands when the data processing framework (DPF) Plugin is loaded. This command can be found in input files automatically generated by the Mechanical Application (see Load Mapping Workflow Specification in the *Mechanical User's Guide*), but manual use of this command is not currently supported.

Argument Descriptions

Action

Action to process. No default, you must choose one of the available options:

WFF --

Load a DPF workflow file.

WFO --

Associate a DPF workflow output pin to an ADPL Variable.

***WorflowId* --**

worflow name used by Mechanical APDL.

Val1

For Action = WFF: DPF workflow filename (typically `WorflowFile.swf`).

For Action = WFO:

APDL Variable name, associated to a DPF output pin.

Notes

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

***PLUG**,*Action*,*PLuginName*

Loads a plugin into Mechanical APDL so that new functionalities can be dynamically added.

DPF commands

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Description

This command can be found in input files automatically generated by the Mechanical Application (see Load Mapping Workflow Specification in the *Mechanical User's Guide*), but manual use of this command is not currently supported.

Argument Descriptions

Action

Action to process. No default, you must choose one of the available options:

LOAD

Load a plugin.

UNLOAD

Unload a plugin to free up memory.

PLuginName

Name (case-sensitive, 32-character maximum) of the plugin to load, which defines the dynamic library name as follows:

- For Windows: *PluginName*.dll.
- For Linux: lib*PluginName*.so

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Commands

A,*P1*,*P2*,*P3*,*P4*,*P5*,*P6*,*P7*,*P8*,*P9*,*P10*,*P11*,*P12*,*P13*,*P14*,*P15*,*P16*,*P17*,*P18*

Defines an area by connecting keypoints.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1, P2, P3, . . . , P18

List of keypoints defining the area (18 maximum if using keyboard entry). At least 3 keypoints must be entered. If *P1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

Notes

Keypoints (*P1* through *P18*) must be input in a clockwise or counterclockwise order around the area. This order also determines the positive normal direction of the area according to the right-hand rule. Existing lines between adjacent keypoints will be used; missing lines are generated "straight" in the active coordinate system and assigned the lowest available numbers (**NUMSTR (p. 1301)**). If more than one line exists between two keypoints, the shorter one will be chosen. If the area is to be defined with more than four keypoints, the required keypoints and lines must lie on a constant coordinate value in the active coordinate system (such as a plane or a cylinder). Areas may be redefined only if not yet attached to a volume. Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>Through KPs

***ABBR**,*Abbr*,*String*

Defines an abbreviation.

APDL (p. 19): Abbreviations (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Abbr

The abbreviation (up to 8 alphanumeric characters) used to represent the string *String*. If *Abbr* is the same as an existing Mechanical APDL command, the abbreviation overrides. Avoid using an *Abbr* which is the same as an Mechanical APDL command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

String

String of characters (60 maximum) represented by *Abbr*. Cannot include a $ or any of the commands **C*** (p. 283)**, **/COM (p. 404)**, **/GOPR (p. 865)**, **/NOPR (p. 1251)**, **/QUIT (p. 1563)**, **/UI (p. 2040)**, or ***END (p. 635)**. Parameter names and commands of the ***DO (p. 567)** and Use the ***IF (p. 927)** groups may not be abbreviated. If *String* is blank, the abbreviation is deleted. To abbreviate multiple commands, create an "unknown command" macro or define *String* to execute a macro file (***USE (p. 2059)**) containing the desired commands.

Notes

Once the abbreviation *Abbr* is defined, you can issue it at the beginning of a command line and follow it with a blank (or with a comma and appended data), and the program will substitute the string *String* for *Abbr* as the line is executed. Up to 100 abbreviations may exist at any time and are available throughout the program. Abbreviations may be redefined or deleted at any time.

Use ***STATUS (p. 1863)** to display the current list of abbreviations. For abbreviations repeated with ***REPEAT (p. 1586)**, substitution occurs before the repeat increments are applied. There are a number of abbreviations that are predefined by the program (these can be deleted by using the blank *String* option described above). Note that *String* will be written to the `File.log`.

This command is valid in any processor.

Menu Paths

Utility Menu>Macro>Edit Abbreviations

Utility Menu>MenuCtrls>Edit Toolbar

AADD, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Adds separate areas to create a single area.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA9

Numbers of areas to be added. If *NA1* = ALL, add all selected areas and ignore *NA2* to *NA9*. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

The areas must be coplanar. The original areas (and their corresponding lines and keypoints) will be deleted by default. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Concatenated entities are not valid with this command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Areas

---

*AFUN, *Lab*

Specifies units for angular functions in parameter expressions.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Lab

Specifies the units to be used:

RAD --

Use radians for input and output of parameter angular functions (default).

DEG --

Use degrees for input and output of parameter angular functions.

STAT --

Show current setting (DEG or RAD) for this command.

Command Default

Use radians for input or output of parameter angular functions.

Notes

Only the SIN, COS, TAN, ASIN, ACOS, ATAN, ATAN2, ANGLEK, and ANGLEN functions (***SET (p. 1720)**, ***VFUN (p. 2106)**) are affected by this command.

Menu Paths

Utility Menu>Parameters>Angular Units

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*ASK, *Par*, *Query*, *DVAL*

Prompts the user to input a parameter value.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

An alphanumeric name used to identify the scalar parameter. See ***SET (p. 1720)** for name restrictions.

Query

Text string to be displayed on the next line as the query (32 characters maximum). Characters having special meaning (such as $! ,) should not be included.

DVAL

Default value assigned to the parameter if the user issues a blank response. May be a number or character string (up to 8 characters enclosed in single quotes). If a default is not assigned, a blank response will delete the parameter.

Notes

Intended primarily for use in macros, the command prints the query (after the word ENTER) on the next line and waits for a response. The response is read from the keyboard, except in batch mode (**/BATCH (p. 217)**), when the response(s) must be the next-read input line(s). The response may be a number, a character string (up to 8 characters enclosed in single quotes), a parameter (numeric or character) or an expression that evaluates to a number. The scalar parameter is then set to the response value. For example, ***ASK (p. 84)**,NN,PARAMETER NN will set NN to the value entered on the next line (after the prompt ENTER PARAMETER NN).

The ***ASK (p. 84)** command is not written to `File.log`, but the responses are written there as follows: If ***ASK (p. 84)** is contained in a macro, the response(s) (only) is written to `File.log` on the line(s) following the macro name. If not contained in a macro, the response is written to `File.log` as a parameter assignment (that is, *Par* = "user-response").

If used within a do-loop that is executed interactively, ***ASK (p. 84)** should be contained in a macro. If not contained in a macro, ***ASK (p. 84)** will still query the user as intended, but the resulting log file will *not* reproduce the effects of the original run.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AATT, *MAT*, *REAL*, *TYPE*, *ESYS*, *SECN*

Associates element attributes with the selected, unmeshed areas.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

The material number to be associated with selected, unmeshed areas.

REAL

The real constant set number to be associated with selected, unmeshed areas.

TYPE

The type number to be associated with selected, unmeshed areas.

ESYS

The coordinate system number to be associated with selected, unmeshed areas.

SECN

The section number to be associated with selected unmeshed areas.

Notes

Areas subsequently generated from the areas will also have these attributes. These element attributes will be used when the areas are meshed. If an area does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT (p. 1111)**, **REAL (p. 1579)**, **TYPE (p. 2036)**, **ESYS (p. 685)**, and **SECNUM (p. 1685)** command settings. Reissue the **AATT (p. 85)** command (before areas are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association. If any of the arguments *MAT*, *REAL*, *TYPE*, *ESYS*, or *SECN* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, Mechanical APDL can proceed with an area meshing operation even when no logical element type has been assigned via **AATT (p. 85)**,,,*TYPE* or **TYPE (p. 2036)**. For more information, see the discussion on setting element attributes in Meshing Your Solid Model in the *Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Areas

Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*AXPY,*vr*,*vi*,*M1*,*wr*,*wi*,*M2*

Performs the matrix operation M2= v*M1 + w*M2.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

vr*,*vi

The real and imaginary parts of the scalar *v*. Default value is 0.

M1

Name of matrix *M1*. If not specified, the operation M2 = w*M2 will be performed.

wr*,*wi

The real and imaginary parts of the scalar *w*. Default value is 0.

M2

Name of matrix *M2*. Must be specified.

Notes

The matrices *M1* and *M2* must have the same dimensions and same type (dense or sparse). If *M2* is real, *vi* and *wi* are ignored.

Menu Paths

This command cannot be accessed from a menu.

ABBRES,*Lab*,*Fname*,*Ext*,*--

Reads abbreviations from a coded file.

APDL (p. 19): Abbreviations (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Lab

Label that specifies the read operation:

NEW --

Replace current abbreviation set with these abbreviations (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CHANGE --

Extend current abbreviation set with these abbreviations, replacing any of the same name that already exist.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

The extension defaults to ABBR if *Fname* is blank.

--

Unused field.

Notes

The abbreviation file may have been written with the **ABBSAV (p. 87)** command. Do not issue **ABBRES (p. 86)**,NEW while inside an executing abbreviation. Doing so will cause all data for the executing abbreviation to be deleted.

This command is valid in any processor.

Menu Paths

Utility Menu>Macro>Restore Abbr

Utility Menu>MenuCtrls>Restore Toolbar

ABBSAV, *Lab*, *Fname*, *Ext*, *--*

Writes the current abbreviation set to a coded file.

APDL (p. 19): Abbreviations (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Lab

Label that specifies the write operation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL --

Write all abbreviations (default).

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

The extension defaults to ABBR if *Fname* is blank.

--

Unused field.

Notes

Existing abbreviations on this file, if any, will be overwritten. The abbreviation file may be read with the **ABBRES (p. 86)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>Macro>Save Abbr

Utility Menu>MenuCtrls>Save Toolbar

ABEXTRACT,*MODE1*,*MODE2*

Extracts the alpha-beta damping multipliers for Rayleigh damping.

`SOLUTION` (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MODE1

First mode number.

MODE2

Second mode number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

ABEXTRACT (p. 88) calls the command macro **DMPEXT (p. 557)** to extract the damping ratio of *MODE1* and *MODE2* and then computes the Alpha and Beta damping multipliers for use in a subsequent structural harmonic or transient analysis. See Damping in the *Structural Analysis Guide* for more information on the alpha and beta damping multipliers. The damping multipliers are stored in parameters ALPHADMP and BETADMP and can be applied using the **ALPHAD (p. 115)** and **BETAD (p. 220)** commands. Before calling **ABEXTRACT (p. 88)**, you must issue **RMFLVEC (p. 1612)** to extract the modal displacements. In addition, a node component FLUN must exist from all FLUID136 nodes. See Introduction for more information on thin film analyses.

This command is also valid in PREP7.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

Main Menu>Solution>ThinFilm>RayleighDamp

ABS, *IR*, *IA*, - -, - -, *Name*, - -, - -, *FACTA*

Forms the absolute value of a variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

- -, - -

Unused fields.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

- -, - -

Unused fields.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FACTA

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

The new variable is calculated as:

IR = | FACTA x IA |

For a complex number (a + *i*b), the absolute value is the magnitude, where the *IA* values are obtained from:

$$\sqrt{a^2 + b^2}$$

See POST26 - Data Operations in the *Mechanical APDL Theory Reference* for details.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Absolute Value

ACCAT, *NA1*, *NA2*

Concatenates multiple areas in preparation for mapped meshing.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2

Areas to be concatenated. If *NA1* = ALL, *NA2* will be ignored and all selected areas (**ASEL (p. 192)**) will be concatenated. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* is ignored).

Notes

Concatenates multiple, adjacent areas (the input areas) into one area (the output area) in preparation for mapped meshing. A volume that contains too many areas for mapped meshing can still be mapped meshed if some of the areas in that volume are first concatenated (see Meshing Your Solid Model in the *Modeling and Meshing Guide* for details on mapped meshing restrictions).

Because of modeling restrictions that result from its use, **ACCAT (p. 90)** is meant to be used solely for meshing. Specifically, (a) the output area and any volumes that have the output area on their area list (**VLIST (p. 2137)**) cannot be used as input to any other solid modeling operation (not even another **ACCAT (p. 90)** command); and (b) the output area cannot accept solid model boundary conditions (**DA (p. 487)**, **SFA (p. 1739)**).

The output area (or volumes which contain it) will be meshed (**AMESH (p. 121)**, **VMESH (p. 2140)**) by meshing the input areas, which themselves must be meshable. The output area from the **ACCAT (p. 90)** operation will be coincident with the input areas and the input areas will be retained. Consider the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AADD (p. 82) command instead of **ACCAT (p. 90)** if you wish to delete the input areas. When an **ACCAT (p. 90)** command is issued, volume area lists (**VLIST (p. 2137)**) that contain *all* of the input areas will be updated so that the volume area lists refer to the output area instead of the input area. Deletion of the output area (**ADELE (p. 97)**) effectively reverses the **ACCAT (p. 90)** operation and restores volume area lists to their original condition. **ACCAT (p. 90)** operations on pairs of adjacent four-sided areas automatically concatenate appropriate lines (**LCCAT (p. 1020)**); in all other situations, line concatenations must be addressed by the user.

You can use the **ASEL (p. 192)** command to select areas that were created by concatenation, and then follow it with an **ADELE (p. 97)**,ALL command to delete them. See Meshing Your Solid Model in the *Modeling and Meshing Guide* for a discussion on how to easily select and delete concatenated areas in one step.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Concatenate>Areas

---

ACCOPTION,*Activate*

Specifies GPU accelerator capability options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Activate

Activates the GPU accelerator capability within the equation solvers.

OFF --

Do not use GPU accelerator.

ON --

Use GPU accelerator.

Command Default

Activate is set to ON when the GPU acceleration command line option (-acc) is specified upon launching Mechanical APDL; otherwise it is OFF.

Notes

The GPU accelerator capability requires specific hardware to be installed on the machine. See the appropriate Ansys, Inc. Installation Guide (Windows or Linux) for a list of supported GPU hardware. Use of this capability also requires HPC licensing. For more information, see GPU Accelerator Capability in the *Parallel Processing Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The GPU accelerator capability is available for the sparse direct solver and the PCG and JCG iterative solvers. Static, buckling, modal, full harmonic, and full transient analyses are supported. For buckling analyses, the Block Lanczos and Subspace eigensolvers are supported. For modal analyses, only the Block Lanczos, PCG Lanczos, Subspace, Unsymmetric, and Damped eigensolvers are supported. Activating this capability when using other equation solvers or other analysis types has no effect.

The GPU accelerator capability is supported only on the Windows 64-bit and Linux 64-bit platforms.

Menu Paths

This command cannot be accessed from a menu.

ACEL,*ACEL_X*,*ACEL_Y*,*ACEL_Z*

Specifies the linear acceleration of the global Cartesian reference frame for the analysis.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ACEL_X, ACEL_Y, ACEL_Z

Linear acceleration of the reference frame along global Cartesian X, Y, and Z axes, respectively.

Notes

In the absence of any other loads or supports, the acceleration of the structure in each of the global Cartesian (X, Y, and Z) axes would be equal in magnitude but opposite in sign to that applied in the **ACEL (p. 92)** command. Thus, to simulate gravity (by using inertial effects), accelerate the reference frame with an **ACEL (p. 92)** command in the direction opposite to gravity.

You can define the acceleration for the following analyses types:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC), full, VT [1], Krylov [1], or mode-superposition [2]
- Transient (**ANTYPE (p. 162)**,TRANS), full or mode-superposition [2]
- Substructure (**ANTYPE (p. 162)**,SUBSTR).

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For all transient dynamic (**ANTYPE (p. 162)**,TRANS) analyses, accelerations are combined with the element mass matrices to form a body-force load vector term. The element mass matrix may be formed from a mass input constant or from a nonzero density (DENS) property, depending upon the element type.

For analysis type **ANTYPE (p. 162)**,HARMIC, the acceleration is assumed to be the real component with a zero imaginary component.

Units of acceleration and mass must be consistent to give a product of force units.

The **ACEL (p. 92)** command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *ACEL_X*, *ACEL_Y*, and *ACEL_Z* input values (***DIM (p. 530)**) as a function of both time and frequency for full transient and harmonic analyses.

Related commands for rotational effects are **CMACEL (p. 358)**, **CGLOC (p. 319)**, **CGOMGA (p. 320)**, **DCGOMG (p. 501)**, **DOMEGA (p. 571)**, **OMEGA (p. 1324)**, **CMOMEGA (p. 368)**, and **CMDOMEGA (p. 362)**.

See Analysis Tools in the *Theory Reference* for more information.

This command is also valid in **/PREP7 (p. 1479)**.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Gravity>Global

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Gravity

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Gravity>Global

Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Gravity

---

ACLEAR, *NA1*, *NA2*, *NINC*

Deletes nodes and area elements associated with selected areas.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

Delete mesh for areas *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the mesh for all selected areas (**ASEL (p. 192)**) is deleted. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

Notes

Deletes *all* nodes and area elements associated with selected areas (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed areas and nodes associated with non-area elements will not be deleted. Attributes assigned as a result of **AATT (p. 85)** are maintained. In the program's response to the command, if an area, line, or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid for rezoning. When issued during rezoning (after the **REMESH (p. 1581)**,START command and before the **REMESH (p. 1581)**,FINISH command), **ACLEAR (p. 93)** clears only the area generated by the **AREMESH (p. 180)** command.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Areas

ADAMS, *NMODES*, *KSTRESS*, *KSHELL*

Performs solutions and writes flexible body information to a modal neutral file (`Jobname.MNF`) for use in an ADAMS analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NMODES

Number of normal modes to be written to `Jobname.MNF` file (no default).

KSTRESS

Specifies whether to write stress or strain results:

0

Do not write stress or strain results (default).

1

Write stress results.

2

Write strain results.

3

Write both stress and strain results.

KSHELL

Shell element output location. This option is valid only for shell elements.

0, 1

Shell top surface (default).

2

Shell middle surface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3

Shell bottom surface.

Notes

ADAMS (p. 94) invokes a predefined Mechanical APDL macro that solves a series of analyses and then writes the modal neutral file, `Jobname.MNF`. This file can be imported into the ADAMS program in order to perform a rigid body dynamics simulation. For detailed information about how to use the **ADAMS (p. 94)** command macro to create a modal neutral file, see Rigid Body Dynamics and the Ansys-Adams Interface in the *Substructuring Analysis Guide*.

Before running the **ADAMS (p. 94)** command macro, you must specify the units with the **/UNITS (p. 2054)** command. The interface points should be the only selected nodes when the command macro is initiated. (Interface points are nodes where constraints may be applied in ADAMS.) Only selected elements will be considered in the calculations.

By default, stress and strain data is transferred to the ADAMS program for all nodes, as specified by the *KSTRESS* value. If you want to transfer stress/strain data for only a subset of nodes, select the desired subset and create a node component named STRESS before running the **ADAMS (p. 94)** command macro. For example, you may want to select exterior nodes for the purpose of visualization in the ADAMS program.

The default filename for the modal neutral file is `Jobname.MNF`. In interactive (GUI) mode, you can specify a filename other than `Jobname.MNF`. In batch mode, there is no option to change the filename, and the modal neutral file is always written to `Jobname.MNF`.

Menu Paths

Main Menu>Solution>ADAMS Connection>Export to ADAMS

ADD, *IR*, *IA*, *IB*, *IC*, *Name*, - -, - -, *FACTA*, *FACTB*, *FACTC*

Adds (sums) variables.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, IB, IC

Reference numbers of the three variables to be operated on. If only two variables, leave *IC* blank. If only one, leave *IB* and *IC* blank.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--,--

Unused fields.

FACTA, FACTB, FACTC

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Adds variables (up to three at once) according to the operation:

$$IR = (FACTA \times IA) + (FACTB \times IB) + (FACTC \times IC)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Add

ADDAM, *AF*, *AA*, *AB*, *AC*, *AD*, *AMIN*

Specifies the acceleration spectrum computation constants for the analysis of shock resistance of shipboard structures.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AF

Direction-dependent acceleration coefficient for elastic or elastic-plastic analysis option (default = 0).

AA, AB, AC, AD

Coefficients for the DDAM acceleration spectrum equations. Default for these coefficients is zero.

AMIN

Minimum acceleration value. It defaults to 6g, where g is the acceleration due to gravity.

Notes

This command specifies acceleration coefficients to analyze shock resistance of shipboard equipment. These coefficients are used to compute mode coefficients according to the equations given in Dynamic Design Analysis Method in the *Mechanical APDL Theory Reference*. The form of these equations is based on the Naval NRL Dynamic Design Analysis Method. This command, along with the **VDDAM (p. 2081)**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and **SED (p. 1704)** commands, is used with the spectrum (**ANTYPE (p. 162)**,SPECTR) analysis as a special purpose alternative to the **SV (p. 1886)**, **FREQ (p. 776)**, and **SVTYP (p. 1887)** commands.

In order to perform a DDAM spectrum analysis using a units system other than BIN (default), you must specify the units system complying with the mass and length units of the model using the **/UNITS (p. 2054)** command. Issue the **/UNITS (p. 2054)** command before defining the shock spectrum computation constants (**ADDAM (p. 96)**). The **ADDAM (p. 96)** command is not supported with the user-defined unite system (*Label* = USER on **/UNITS (p. 2054)**).

DDASPEC (p. 503) may alternatively be used to calculate spectrum coefficients.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options

Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options

ADELE, *NA1*, *NA2*, *NINC*, *KSWP*

Deletes unmeshed areas.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

Delete areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas (**ASEL (p. 192)**) are deleted. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

KSWP

Specifies whether keypoints and lines are also to be deleted:

0

Delete areas only (default).

1

Delete areas, as well as keypoints and lines attached to specified areas but not shared by other areas.

Notes

An area attached to a volume cannot be deleted unless the volume is first deleted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Area and Below

Main Menu>Preprocessor>Modeling>Delete>Areas Only

ADGL,*NA1*,*NA2*,*NINC*

Lists keypoints of an area that lie on a parametric degeneracy.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

List keypoints that lie on a parametric degeneracy on areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* will be ignored and keypoints on all selected areas (**ASEL (p. 192)**) will be listed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted in *NA1* (*NA2* and *NINC* will be ignored).

Notes

See the *Modeling and Meshing Guide* for details on parametric degeneracies.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>List Degen Areas

Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>List Degen Areas

---

ADPCI,*Action*,*Par1*,*Par2*,*Par3*,*Par4*,*Par5*,*Par6*,*Par7*,*Par8*

Defines parameters associated with adaptive crack initiation.

PREP7 (p. 22): Special Purpose (p. 37)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Specifies action for defining or manipulating initial crack data:

DEFINE

Initiate a new adaptive-crack calculation and assign an ID. (p. 99)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GEOM

Define the geometry data for initializing a crack. (p. 99)

DELE

Delete the **ADPCI (p. 98)** data set associated with the specified ID. (p. 101)

LIST

List the **ADPCI (p. 98)** data set associated with the specified ID. (p. 101)

Command Specification for *Action* = DEFINE

ADPCI (p. 98), DEFINE, *Par1*, *Par2*, *Par3*, *Par4*

Par1 --

ADPCI (p. 98) ID number.

Par2 --

Node component name ($\leq$ 32 characters) for defining the crack-initiation zone.

Par3 --

Material ID for defining the crack-initiation criterion.

Par4 --

Shape of the initialized crack:

ELLIPSE – Elliptical crack shape (default, and the only valid value).

Command Specifications for *Action* = GEOM

ADPCI (p. 98), GEOM, *Par1*, *Par2*, *Par3*, *Par4*, *Par5*, *Par6*, *Par7*, *Par8*

Par1 --

ADPCI (p. 98) ID number.

Par2 --

Crack geometry characteristic to define:

CENTER – Specify the ellipse center location.

AXES – Specify the directions of the two ellipse axes.

ALEN – Specify the lengths of two ellipse axes. This value is required when AXES or LCS is specified.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LCS – Local coordinate system number for defining the ellipse center location and axes directions. See "Notes" (p. 101) below.

Par3 --

The first value to assign to the geometry characteristic specified via *Par2*:

If *Par2* = CENTER, the X coordinate of the ellipse center.

If *Par2* = AXES, the X component of the first ellipse axis direction.

If *Par2* = ALEN, the length of the first ellipse axis.

If *Par2* = LCS, the local coordinate system number.

Par4 --

The second value to assign to the geometry characteristic specified via *Par2*:

If *Par2* = CENTER, the Y coordinate of the ellipse center.

If *Par2* = AXES, the Y component of the first ellipse axis direction.

If *Par2* = ALEN, the length of the second ellipse axis.

Par5 --

The third value to assign to the geometry characteristic specified via *Par2*:

If *Par2* = CENTER, the Z coordinate of the ellipse center.

If *Par2* = AXES, the Z component of the first ellipse axis direction.

Par6 --

The fourth value to assign to the geometry characteristic specified via *Par2*:

If *Par2* = AXES, the X component of the second ellipse axis direction.

Par7 --

The fifth value to assign to the geometry characteristic specified via *Par2*:

If *Par2* = AXES, the Y component of the second ellipse axis direction.

Par8 --

The sixth value to assign to the geometry characteristic specified via *Par2*:

If *Par2* = AXES, the Z component of the second ellipse axis direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specifications for *Action* = DELE

ADPCI (p. 98), DELE, *Par1*

Par1 --

ADPCI (p. 98) ID (default = ALL).

Command Specifications for *Action* = LIST

ADPCI (p. 98), LIST, *Par1*

Par1 --

ADPCI (p. 98) ID number (default = ALL).

Notes

For **ADPCI (p. 98)**,GEOM,LCS, the ellipse center locates at the origin of the local coordinate system. The local coordinate system Y axis defines the plane normal of the ellipse, and X and Z axes define two orientations of the ellipse. (The LCS argument is equivalent to combining the CENTER and AXES arguments. Separate **ADPCI (p. 98)**,GEOM commands to specify those arguments are therefore not issued.)

For more information about using **ADPCI (p. 98)** in a crack-initiation analysis, see SMART Method for Crack-Initiation Simulation in the *Fracture Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

ADRAG, *NL1, NL2, NL3, NL4, NL5, NL6, NLP1, NLP2, NLP3, NLP4, NLP5, NLP6*

Generates areas by dragging a line pattern along a path.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NL3, . . . , NL6

List of lines in the pattern to be dragged (6 maximum if using keyboard entry). Lines should form a continuous pattern (no more than two lines connected to any one keypoint. If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *NL1* = ALL, all selected lines (except those that define the drag path) will be swept along the path. A component name may also be substituted for *NL1*.

NLP1, NLP2, NLP3, . . . , NLP6

List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates areas (and their corresponding keypoints and lines) by sweeping a given line pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NLP1*, *NLP2*, etc.). If the drag path is a single line (*NLP1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given line pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves.

Keypoint, line, and area numbers are automatically assigned, beginning with the lowest available values (**NUMSTR (p. 1301)**). Adjacent lines use a common keypoint. Adjacent areas use a common line. For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Lines>Along Lines

---

AEROCOEFF,*AeroModeType*,*AeroMappedFileNames*,*AeroSpecs*,*AeroScalar*,*nBlades*,*AutoFileRead*

Computes the aero-damping and stiffness coefficients and writes them to an APDL array.

PREP7 (p. 22): Special Purpose (p. 37)
SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AeroModeType

Mode type to be used.

BLADE

Non-cyclic cantilevered blade mode (default)

AeroMappedFiles

Name of string array containing file names of mapped pressures from CFD. The file names should be ordered to correspond to the *AeroSpecs* array.

AeroSpecs

Name of numerical array containing data organized to correspond to the *AeroMappedFiles* array. See the "Notes" (p. 103) section for specific information that must be in the array.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AeroScalar

Scaling value(s) to handle any modal scaling difference between structural and CFD modes. The values can be entered as a scalar or 1-dimensional array. (each scaling value defaults to 1)

nBlades

Number of blades.

AutoFileRead

Key to automatically read and use values from CFD file header.

0 (OFF or NO)

Do not read scaling values or nodal diameter from the CFD file header. (default)

1 (ON or YES)

Read scaling values (labeled `Mode Multiplier` in CFD file) from CFD file header. The scaling values read will be used in calculations and the *AeroScalar* input will be ignored. The nodal diameter values will be used to cross check the value of i (input through *AeroSpecs* array).

Command Default

No defaults are available for the **AEROCOEFF (p. 102)** command.

Notes

The **AEROCOEFF (p. 102)** command is designed to generate an array of aerodynamic coefficients that can be used in a cyclic mode-superposition harmonic response analysis using the **CYCFREQ (p. 458)**,AERO command to represent aerodynamic stiffness and damping. These aerodynamic coefficients can also be used in a damped modal analysis phase (**CYCFREQ (p. 458)**,MODAL) of a cyclic mode-superposition harmonic solve. An APDL array called *Jobname*AeroArray is generated using the **AEROCOEFF (p. 102)** command. This array is compatible with the array needed for the **CYCFREQ (p. 458)**,AERO command.

The format of the written array follows that of the **CYCFREQ (p. 458)**,AERO command. The array is formatted as follows:

$$i, m, n, V_{\text{real}}, V_{\text{imag}}$$

where

i = the ith interblade phase angle (IBPA)

m = the mth vibrating blade mode

n = the nth blade mode generating the pressure oscillations

V_{real} and V_{imag} = the real and imaginary coefficients.

Prior to issuing the **AEROCOEFF (p. 102)** command, a non-cyclic cantilevered blade modal analysis must be run, either stress-free or prestressed using linear perturbation. For more information, see Modal

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis in the *Structural Analysis Guide*. The file requirements for the **AEROCOEFF (p. 102)** command are the same as those needed for modal restart as described in Modal Analysis Restart.

The *AeroSpecs* values are specified in a 3×r array (***DIM (p. 530)**), where r is a positive integer equal to the number of interblade phase angles and the pressure modes solved for in the CFD analysis. Each row has the structure:

$$i,m,n$$

where

i = the ith interblade phase angle (IBPA)

m = the mth vibrating blade mode

n = the nth blade mode generating the pressure oscillations

At least one aerodynamic damping coefficient must be specified for each IBPA (equal to the number of blades) while keeping m and n constant. If a value is not specified, the program writes an array value of zero for both V_{real} and V_{imag}. The values of m and n are relative to the modes computed in the required modal analysis.

The number of *AeroScalar* values must be equal to the number of pressure modes (n from *AeroSpecs*). If the number of *AeroScalar* values is greater than 1, the values must be entered by defining an n×1 array (***DIM (p. 530)**) and entering the array name in the *AeroScalar* field. For a discussion of how *AeroScalar* values are computed, see Scaling Aerodynamic Coupling Coefficients.

The value for *nBlades* should be equal to the number of sectors of the system. If there are multiple blades per cyclic sector, then the combination of blades on the single sector will have an aero coefficient value. In this case, each blade will not have a distinct aero coefficient.

Menu Paths

This command cannot be accessed from a menu.

AESIZE, *ANUM*, *SIZE*,

Specifies the element size to be meshed onto areas.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ANUM

Area number of the area to which this element size specification applies. If *ANUM* = ALL, size applies to all selected areas. If *ANUM* = P, graphical picking is enabled. A component name may also be substituted for *ANUM*.

SIZE

Desired element size.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

AESIZE (p. 104) allows control over the element sizing inside any area or on the face(s) of a volume.

SIZE controls element size on the interior of the area. For any line on the area not having its own size assignment and not controlled by keypoint size assignments, it specifies the element size along the line as well, so long as no adjacent area has a smaller size, which would take precedence. If the **AESIZE (p. 104)** governs the boundary and SmartSizing is on, the boundary size can be refined for curvature or proximity.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>All Areas

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>Clr Size

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Areas>Picked Areas

---

AFILLT, *NA1*, *NA2*, *RAD*

Generates a fillet at the intersection of two areas.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1

Number of the first intersecting area. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

NA2

Number of the second intersecting area.

RAD

Radius of fillet to be generated.

Notes

Generates an area of constant fillet radius at the intersection of two areas using a series of Boolean operations. Corresponding lines and keypoints are also generated. See **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. If areas do not initially intersect at a common line, use the **AINA (p. 109)** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Area Fillet

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AFLIST

Lists the current data in the database.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Lists the current data and specifications in the database. If batch, lists all appropriate data. If interactive, lists only summaries.

Menu Paths

Utility Menu>List>Other>Database Summary

AFSURF, *SAREA*, *TLINE*

Generates surface elements overlaid on the surface of existing solid elements and assigns the extra node as the closest fluid element node.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SAREA

Component name for the surface areas of the meshed solid volumes. The component name must be 32 characters or less.

TLINE

Component name for the target lines meshed with fluid elements. The component name must be 32 characters or less.

Notes

This command macro is used to generate surface effect elements overlaid on the surface of existing solid elements and, based on proximity, to determine and assign the extra node for each surface element. The underlying volumes of the solid region and the fluid lines must be meshed prior to calling this command macro. The active element type must be SURF152 with appropriate settings for KEYOPT(4), KEYOPT(5), KEYOPT(6), and KEYOPT(8).

The surface areas of the solid and the target lines of the fluid are grouped into components and named using the **CM (p. 356)** command. The names must be enclosed in single quotes (for example, '*SAREA*') when the **AFSURF (p. 106)** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the SURF152 and FLUID116 element types.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid

---

AGEN, *ITIME*, *NA1*, *NA2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional areas from a pattern of areas.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be more than 1 for generation to occur.

NA1, NA2, NINC

Generate areas from the pattern of areas *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas (**ASEL (p. 192)**). If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

DX, DY, DZ

Keypoint location increments in the active coordinate system (--, D θ, DZ for cylindrical; --, D θ, -- for spherical).

KINC

Keypoint number increment between generated sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**}.

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original areas, if they exist.

1

Do not generate nodes and elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IMOVE

Specifies whether to redefine the existing areas:

0

Generate new areas as requested with the *ITIME* argument.

1

Move original areas to new position, retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). If the original areas are needed in the original position (for example, they may be attached to a volume), they are not moved, and new areas are generated instead. Meshed items corresponding to moved areas are also moved if not needed at their original position.

Notes

Generates additional areas (and their corresponding keypoints, lines and mesh) from a given area pattern. The MAT, TYPE, REAL, ESYS, and SECNUM attributes of the new areas are based upon the areas in the pattern and not upon the current settings of the pointers. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial. Generations which produce areas of a size or shape different from the pattern (that is, radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Solid modeling in a toroidal coordinate system is not recommended. Area and line numbers are automatically assigned, beginning with the lowest available values (**NUMSTR (p. 1301)**).

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Areas

Main Menu>Preprocessor>Modeling>Move / Modify>Areas>Areas

AGLUE, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Generates new areas by "gluing" areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA9

Numbers of the areas to be glued. If *NA1* = ALL, all selected areas will be glued (*NA2* to *NA9* will be ignored). If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

Use of the **AGLUE (p. 108)** command generates new areas by "gluing" input areas. The glue operation redefines the input areas so that they share lines along their common boundaries. The new areas en-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

compass the same geometry as the original areas. This operation is only valid if the intersection of the input areas are lines along the boundaries of those areas. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to new entities generated.

The **AGLUE (p. 108)** command results in the merging of lines and keypoints at the common area boundaries. The lines and keypoints of the lower numbered area will be kept. This means one must be aware of area numbering when multiple **AGLUE (p. 108)** commands are applied to avoid any "ungluing" of geometry.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Areas

AINA, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Finds the intersection of areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA9

Numbers of areas to be intersected. If *NA1* = ALL, *NA2* to *NA9* are ignored and the intersection of all selected areas is found. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

Finds the common (not pairwise) intersection of areas. The common intersection is defined as the regions shared (in common) by **all** areas listed on this command. New areas will be generated where the original areas intersect. If the regions of intersection are only lines, new lines will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AINP, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Finds the pairwise intersection of areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA9

Numbers of areas to be intersected pairwise. If *NA1* = ALL, *NA2* to *NA9* are ignored and the pairwise intersection of all selected areas is found. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted for *NA1*.

Notes

Finds the pairwise intersection of areas. The pairwise intersection is defined as all regions shared by any two or more areas listed on this command. New areas will be generated where the original areas intersect pairwise. If the regions of pairwise intersection are only lines, new lines will be generated. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Areas

AINV, *NA*, *NV*

Finds the intersection of an area with a volume.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA

Number of area to be intersected. If P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

NV

Number of volume to be intersected.

Notes

New areas will be generated where the areas intersect the volumes. If the regions of intersection are only lines, new lines will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Area with Volume

---

AIRL, *NRB*, *RIGID_CALC*

Specifies that automatic inertia relief calculations are to be performed.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NRB

Number of rigid body modes in the model:

AUTO

Activate automatic inertia relief (default). The program automatically determines the number of rigid body modes.

n

Activate automatic inertia relief with the assumption of *n* rigid body modes ($1 \leq n \leq 6$).

0

Deactivate automatic inertia relief.

RIGID_CALC

Method for computing the rigid body modes:

0

Use an eigensolver to compute rigid body modes (default).

1

Use the geometry to compute rigid body modes (valid only when *NRB* = AUTO). The geometric rigid body vectors are calculated according to Equation 15.171 in Participation Factors and Mode Coefficients in the *Theory Reference*.

Command Default

No automatic inertia relief calculations.

Notes

The **AIRL (p. 111)** command activates automatic inertia relief for models having up to six rigid body modes. This method is only valid for linear static analyses (**ANTYPE (p. 162)**,STATIC).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default (*NRB* = AUTO), the number of rigid body modes in the structure is automatically determined. The static solution is altered so that the inertial effects factor into counterbalancing the external loads. This method relies on calculation of the rigid body modes using either an eigensolver (*RIGID_CALC* = 0, which is the default) or the model geometry (*RIGID_CALC* = 1).

For the *NRB* = AUTO option, no supports should be defined.

For a model that is partially constrained by design, you must set *NRB* to the number of rigid body modes present in the structure and set *RIGID_CALC* = 0. The use of geometry (*RIGID_CALC* = 1) to compute the rigid-body modes of a partially constrained model is not supported.

Loads may be input as usual. Displacements and stresses are calculated as usual.

Use **IRLIST (p. 962)** to print inertia relief calculation results. The mass and moment of inertia summary printed before the solution is accurate (because of the additional pre-calculations required for inertia relief). See Inertia Relief in the *Theory Reference* for calculation details. See Including Inertia Relief Calculations in the *Basic Analysis Guide* for procedural details, including requirements and limitations.

This command is also valid in PREP7.

Example Usage

Example command input for including automatic inertia relief calculations in a linear static analysis is found in Including Inertia Relief Calculations in the *Basic Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

AL, *L1*, *L2*, *L3*, *L4*, *L5*, *L6*, *L7*, *L8*, *L9*, *L10*

Generates an area bounded by previously defined lines.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

L1, L2, L3, . . . , L10

List of lines defining area. The minimum number of lines is 3. The positive normal of the area is controlled by the direction of *L1* using the right-hand rule. A negative value of *L1* reverses the normal direction. If *L1* = ALL, use all selected lines with *L2* defining the normal (*L3* to *L10* are ignored and *L2* defaults to the lowest numbered selected line). If *L1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *L1*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Lines may be input (once each) in any order and must form a simply connected closed curve. If the area is defined with more than four lines, the lines must also lie in the same plane or on a constant coordinate value in the active coordinate system (such as a plane or a cylinder).

Note:

Solid modeling in a toroidal coordinate system is not recommended. Areas may be re-defined only if not yet attached to a volume.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Lines

---

ALIST, *NA1*, *NA2*, *NINC*, *Lab*

Lists the defined areas.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

List areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas (**ASEL (p. 192)**) are listed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

Lab

Determines what type of listing is used (one of the following):

(blank)

Prints information about all areas in the specified range.

HPT

Prints information about only those areas that contain hard points.

Notes

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **AATT (p. 85)** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer (**TYPE (p. 2036)**, **MAT (p. 1111)**, **REAL (p. 1579)**, or **ESYS (p. 685)**) that was active during meshing (and will be reset to zero if the mesh is cleared). A "-1" in the "nodes" column indicates that

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the area has been meshed but there are no interior nodes. The area size is listed only if an **ASUM (p. 203)** command has been performed on the area.

Menu Paths

Utility Menu>List>Areas

ALLSEL, *LabT*, *Entity*

Selects all entities with a single command.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabT

Type of selection to be made:

ALL

Selects all items of the specified entity type and all items of lower entity types (default).

BELOW

Selects all items directly associated with and below the selected items of the specified entity type.

Entity

Entity type on which selection is based:

ALL

All entity types (default).

VOLU

Volumes.

AREA

Areas.

LINE

Lines.

KP

Keypoints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ELEM

Elements.

NODE

Nodes.

Notes

ALLSEL (p. 114) is a convenience command that allows the user to select all items of a specified entity type or to select items associated with the selected items of a higher entity.

An entity hierarchy is used to decide what entities will be available in the selection process. This hierarchy from top to bottom is as follows: volumes, areas, lines, keypoints, elements, and nodes. The hierarchy may also be divided into two branches: the solid model and the finite element model. The label ALL selects items based on one branch only, while BELOW uses the entire entity hierarchy. For example, **ALLSEL (p. 114)**,ALL,VOLU selects all volumes, areas, lines, and keypoints in the data base. **ALLSEL (p. 114)**,BELOW,AREA selects all lines belonging to the selected areas; all keypoints belonging to those lines; all elements belonging to those areas, lines, and keypoints; and all nodes belonging to those elements.

The $ character should not be used after the **ALLSEL (p. 114)** command.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking
Utility Menu>Select>Everything
Utility Menu>Select>Everything Below>Selected Areas
Utility Menu>Select>Everything Below>Selected Elements
Utility Menu>Select>Everything Below>Selected Keypoints
Utility Menu>Select>Everything Below>Selected Lines
Utility Menu>Select>Everything Below>Selected Volumes

ALPHAD, *VALUE*

Defines the mass matrix multiplier for damping.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VALUE

Mass matrix multiplier for damping.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command defines the mass matrix multiplier α used to form the viscous damping matrix $[C]=\alpha[M]$, where $[M]$ is the mass matrix.

Values of α can also be input as a material property (**MP (p. 1160)**,ALPD or **TB (p. 1899)**,SDAMP,,,,ALPD). If ALPD in either form is included, the ALPD value is added to the **ALPHAD (p. 115)** value as appropriate. (See Material Damping in the *Material Reference* and Damping Matrices in the *Theory Reference*.) Damping is not used in static (**ANTYPE (p. 162)**,STATIC) or buckling (**ANTYPE (p. 162)**,BUCKLE) analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping

Main Menu>Solution>Analysis Type>Sol'n Controls>Transient

Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

AMAP, *AREA*, *KP1*, *KP2*, *KP3*, *KP4*

Generates a 2D mapped mesh based on specified area corners.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Area number of area to be meshed. If *AREA* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

KP1, KP2, KP3, KP4

Keypoints defining corners of the mapped mesh. Three or four corners may be specified, and may be input in any order.

Notes

Only one area at a time can be meshed with this command. The program *internally* concatenates all lines between the specified keypoints, then meshes the area with all quadrilateral elements. If line divisions are set, the mesh will follow the rules for mapped meshing (see Meshing Your Solid Model in the *Modeling and Meshing Guide*).

If the area being meshed has concatenated lines, the program will ask if those concatenations should be removed (in batch, the concatenations will automatically be removed). Nodes required for the generated elements are created and assigned the lowest available node numbers. If a mapped mesh is not possible due to mismatched line divisions or poor element shapes, the meshing operation is aborted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>By Corners

AMBEAM, *NUMBEAMS*, *POWER*, *ABSORPTIVITY*

Specifies energy source parameters in an additive manufacturing analysis.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

NUMBEAMS

Number of beams used in the build process. Default = 1.

POWER

(Optional) Beam power for the build process.

ABSORPTIVITY

(Optional) Beam absorptivity value. Must be between 0 and 1. Default = 1.

Command Default

One beam is used in the build process.

If *POWER* is not specified, the melting temperature will be used for heating steps.

ABSORPTIVITY is only used if *POWER* is also specified.

Notes

This command is also valid in PREP7.

For the *POWER* option, if a value is specified, then *POWER* * *ABSORPTIVITY* will be used with other process and geometric parameters (scan speed, hatch spacing, deposition thickness, heating time, superlayer volume) to define the heat generation to heat new element layers. The heating time for this option is specified with the **AMBUILD (p. 118)**,TIME command. If a *POWER* is not specified, new layers will be set to the melting temperature (**AMMAT (p. 121)**).

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AMBUILD,*Option*,*VAL1*,*VAL2*,*VAL3*,*VAL4*

Specifies printer parameters for the build and other options in an additive manufacturing analysis.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

Option

Option:

LAYERT

VAL1 – Deposition-layer thickness.

VAL2 – Mesh height.

VAL3 – Error-checking flag. Set to 0 (default) or 1. Setting to 1 causes the application to omit error checks for consistent element size and elements spanning across layers.

SCAN

VAL1 – Hatch spacing.

VAL2 – Beam-travel speed.

TIME

VAL1 – Inter-layer dwell time. Default = 0.0.

VAL2 – Dwell-time multiplier for multiple parts on the build plate or number of repeated symmetry sectors in simulations with symmetry. Default = 1.0.

VAL3 – Heating step time for use with beam power option (**AMBEAM (p. 117)**,,*POWER*).

VAL4 – Scan time table.

PLATE

VAL1 – Z-coordinate of the top of the build plate. Default = 0.0.

CHECK

VAL1 – If YES, create the build-summary file but do not solve. Default = NO.

RTHFILE

VAL1 – Name of the thermal-results file (including its path). Default = `file.rth` in the current working directory.

SSF

VAL1 – Strain Scaling Factor. Default = 1.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

None. An *Option* label is required.

Notes

If using a layered tetrahedral mesh, specify the mesh height (LAYERT,*VAL2*). For Cartesian meshes, the mesh height is determined automatically.

When setting the error-checking flag (LAYERT,,,1), verify your model and results carefully. Using the flag may lead to improper setup of layers or boundary conditions.

The hatch spacing and beam travel speed are the average values used during the build.

The inter-layer dwell time (TIME,*VAL1*) is the span of time from the end of the deposition of a layer to the start of the deposition of the next layer. It includes the time required for recoater-blade repositioning and powder-layer spreading.

The dwell-time multiplier (TIME,,*VAL2*) accounts for more than one part being printed on the build plate, or it is used to reconcile build time in simulations using symmetry. For multiple parts on a build plate, if the additional parts are the same part as the one being simulated and are arranged in the same orientation on the build plate, the multiplier is the total number of parts. If different parts exist on the plate, the multiplier is an estimate of the time required to build the other parts relative to the part being simulated. In simulations with symmetry, the dwell-time multiplier is the total number of repeated symmetry sectors: 2 for half symmetry, 4 for ¼ symmetry, and so on.

The heating step time (TIME,,,*VAL3*) is used along with other defined process and geometric parameters (power, scan speed, hatch spacing, deposition thickness, superlayer volume) to set the time duration and magnitude of heat generation on new element layers. A value of 0 (default) sets the heating time to the minimum time step constraint. See General Options in the *Thermal Analysis Guide*. When power has been specified, you can alternatively enter SCAN in this field to set the heating time to the amount of time it would take to scan the volume of material in each element layer. With the SCAN option, the cooling between layers will still be the recoater time. Setting *VAL3* = SCAN may give better temperature results at the end of the layer and is more stable across different mesh sizes. Note that the SCAN option will have a different end time of the simulation because the layer thickness adjustment for cooling will not be used.

The scan time (TIME,*VAL4*) represents the amount of time it takes to scan a real layer. By default, the scan time will be determined from each layer's cross-sectional area and other process parameters. When specified, it must be defined as a table with times specified on the Z primary variable. Times that are averaged or interpolated from the table should not include recoating time and will be adjusted to account for superlayer size compared to the deposition thickness.

When specifying the name of the thermal-results file (RTHFILE,*VAL1*), omit the `.rth` extension. The program also looks for the `thermal.build` file in the same path.

The strain scaling factor (SSF,*VAL1*) scales the thermal strains in the structural portion of thermal-structural simulations by the specified value.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Menu Paths

This command cannot be accessed from a menu.

AMENV, *TGAS*, *HGAS*

Specifies the build-environment thermal boundary conditions in an additive manufacturing analysis.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

TGAS

Temperature of the gas in the build enclosure.

HGAS

Convection coefficient from the part to the enclosure gas.

Command Default

No convection is included.

Notes

If using the power-bed fusion process (**AMTYPE (p. 128)**,PBF), the convection is applied only to the top of a newly laid layer.

If using the directed-energy deposition process (**AMTYPE (p. 128)**,DED), the convection is applied to the top of a newly laid layer *and* to the sides of the part already built.

No convection boundary conditions are applied to the plate, although you can define them manually (**SF (p. 1733)** and related commands).

This command is also valid in PREP7.

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AMESH, *NA1*, *NA2*, *NINC*

Generates nodes and area elements within areas.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

Mesh areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas (**ASEL (p. 192)**) are meshed. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

Notes

Any undefined nodes required for the generated elements are created and assigned the lowest available numbers.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Free

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>3 or 4 sided

Main Menu>Preprocessor>Meshing>Mesh>Areas>Target Surf

AMMAT, *MATPART*, *TMELT*, *TRELAX*

Specifies the melting and relaxation temperatures of the build material in an additive manufacturing analysis.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

MATPART

The material ID of the build part. Default = 1.

TMELT

Melting temperature of the build part (required).

TRELAX

Relaxation temperature of the build part (optional).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

The material ID of the build part is 1. A melting temperature is required.

Notes

This command is required in an additive manufacturing analysis.

If the part consists of multiple material IDs, you can specify any of the material IDs (*MATPART*), as all are of the same material.

The melting temperature (*TMELT*) is the temperature at which thermal strains begin to accumulate. This value is typically the liquidus-to-solidus temperature, but may be less for some phase-transition material (such as Ti64).

The relaxation temperature (*TRELAX*) is the temperature at which the strains are zeroed out (annealed). You can use *TRELAX* during the build process (**AMSTEP (p. 124)**,BUILD) to account for stress relaxation, but it serves primarily as a simplified stress-relaxation method during the heat-treat step (**AMSTEP (p. 124)**,HEATTREAT). (A creep model offers a more stringent stress-relaxation approach if needed.)

This command is also valid in PREP7.

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Menu Paths

This command cannot be accessed from a menu.

AMPOWDER, *TPOWDER*, *HPOWDER*, *MATFACTOR*

Specifies the thermal conditions of the powder in an additive manufacturing analysis.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

TPOWDER

Temperature of the newly added powder.

HPOWDER

Effective convection coefficient from the part to the powder bed.

MATFACTOR

Knockdown factor applied to the solid material properties (to obtain the powder material properties). Default = 0.01.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

No heat flow is included into the powder bed.

Notes

This command applies only to the powder-bed fusion (**AMTYPE (p. 128)**,PBF) process.

To estimate the convection coefficient (*HPOWDER*), divide the conduction property of the powder (its KXX) by a characteristic conduction length into the powder (for example, ¼ of the distance from the part boundary to the build-chamber wall).

The program uses the knockdown factor (*MATFACTOR*) to estimate the powder properties. The program applies the factor (typically 0.01) to the solid material properties to estimate the properties of the material in its powder state. The powder-state properties are used during the heating of the new layer (before its subsequent solidification and cooldown) prior to the next layer being applied.

This command is also valid in PREP7.

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Menu Paths

This command cannot be accessed from a menu.

AMRESULT,*Item*,*Key*

Specifies additive manufacturing result data written to a `.txt` file.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

Item

Result item to output to a tab-delimited `.txt` file:

RINT

Recoater interference. Available in a *structural* additive manufacturing analysis only.

DTEMP

Layer end temperature. Available in a *thermal* additive manufacturing analysis only.

HSTN

High Strain. Available in a *structural* additive manufacturing analysis only.

Key

Write-control key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Does not write the specified result item (default).

ON

Writes the specified result item.

Command Default

Result items are not written.

Notes

This command controls additive manufacturing result data written to a `.txt` file. Specifically, `AMResults.txt` is written for recoater interference and layer end temperature, and `AMHighStrain.txt` is written for high strains. The specified results are not written to the database results (`.RST`) file.

Result items written to the `.txt` file also include node numbers and x, y, z locations.

RINT gives the z-deformation of a layer just before a new layer is applied. This result value can help to determine whether an issue may occur when spreading a new layer.

DTEMP gives the temperature of a layer just before a new layer is applied. This result value can help to identify regions where the build may be overheating that may result in problematic thermal conditions.

HSTN gives the maximum equivalent strain experienced during the build process. This result value can help to identify regions at risk of cracking.

Menu Paths

This command cannot be accessed from a menu.

AMSTEP, *Sequence*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*

Specifies the process-sequence steps in an additive manufacturing analysis.

`SOLUTION` (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**
Product Restrictions (p. 126)

Sequence

One of the following sequence options:

PREHEAT

VAL1 – Preheat temperature of the build plate. Default = **TUNIF (p. 2027)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BUILD

VAL1 – Unused.

VAL2 – Ending-layer number (for performing the simulation from the first layer to the specified layer only). Default = last layer necessary to build the part.

VAL3 – Number of time steps taken to apply heating. Default = 2.

VAL4 – Number of time steps taken between layer additions. Default = 2.

VAL5 – Unused.

VAL6 – Bias growth factor for time steps between layer additions. Default = 1.

COOLDOWN

VAL1 – Ambient (room) temperature (to serve as the target cooldown temperature).

VAL2 – Cooldown time. If 0 or unspecified, the program calculates the time based on the volume of the part and the convection coefficient (**AMENV (p. 120)** and **AMPOWDER (p. 122)**).

VAL3 – Number of time steps taken to cool down. Default = 20.

UNBOLT

Perform a base unbolting step where base constraints are removed.

HEATTREAT

Perform a heat-treat (stress-relief) step.

REMOVE

VAL1 – The number of a support to remove. Specify 0 for plate.

VAL2 – Directional cutoff step size. Only valid if *VAL1* = 0.

VAL3 – Directional cutoff angle specified on the X-Y plane from the +X axis. Only valid if *VAL1* = 0. Default = 0 radians.

USER

Perform a user-defined step.

Command Default

None. A sequence option is required.

Notes

AMSTEP (p. 124) executes a process-sequence step:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In a thermal analysis, *Sequence* = PREHEAT sets the value of the starting temperature of the build plate. It is ignored in a structural analysis.
- *Sequence* = BUILD executes the layer-by-layer build sequence.
- *Sequence* = COOLDOWN executes the cooldown step and must occur after the BUILD step.
- *Sequence* = HEATTREAT performs a heat-treatment step to stress-relieve the part. Issue **AMBUILD (p. 118)**,RTHFILE to point to the heat-treat thermal-cycle results and specify either a relaxation temperature (**AMMAT (p. 121)**) or creep properties (**TB (p. 1899)**).
- In a structural analysis, *Sequence* = REMOVE removes the requested support or build plate. It is ignored in a thermal analysis.
- In a structural analysis, *Sequence* = USER can specify an initial step (such as bolt-pretension the build plate) or a final step (such as a manufacturing postprocessing step).

For *Sequence* = BUILD and *Sequence* = COOLDOWN, the number of time steps specified determines the accuracy of the captured temperature profile. For distortion and global residual stresses, the default is usually sufficient. With some materials (Al alloys in particular), the default of evenly spaced time steps during the build (*Sequence* = BUILD) may not adequately capture the cooldown. The bias growth factor (*VAL6*) adjusts the time spacing to better resolve temperatures as they cool between layers.

When *Sequence* = UNBOLT, constraints for UX, UY, and UZ will be removed from base nodes. Constraints on weak spring or nonbuild (NONBUILD_ELEMENTS component name) elements will not be deleted. Similarly, symmetry nodes (SYMM_NODES component name) will not be deleted. Note that for symmetry this may mean that the Z, and other undesired constraints on symmetry nodes may need to be deleted manually for this step.

For *Sequence* = REMOVE, directional cutoff is activated when *VAL1* is set to 0 (plate), and a value is given for *VAL2* (cut step size). This option will sequentially remove the first layer of elements in a series of steps with each step moving the specified distance (*VAL2*). The cutoff steps will continue across the entire part in the direction of the specified angle (*VAL3*). If neither *VAL1* nor *VAL2* are specified, the entire base is removed in a single instantaneous step.

When *Sequence* = USER, the process-sequence steps are bypassed, and the usual nonlinear solution is performed during this step. All applicable load and load step options are accessible. If USER is the initial step, all times are offset by the TIME associated with the USER step.

This command starts a solution. You must remain in SOLUTION between sequence steps.

This command is valid in the SOLUTION processor only.

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Product Restrictions

Ansys Mechanical Enterprise PrepPost

This command is not valid.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

AMSUPPORTS, *NSUPPORTS*, *CompName*, *SectArray*

Specifies information about the supports in an additive manufacturing analysis.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

NSUPPORTS

Number of supports.

CompName

Root name of the components containing the elements comprising each support. (For example, if *CompName* = "MySupport," MySupport1 represents support 1, MySupport2 represents support 2, etc.)

SectArray

Name of the array (***DIM (p. 530)**) containing the section-reference ID for each support.

Command Default

No supports are included in the build. All non-plate elements are assumed to be the build part.

Notes

The section-reference ID array (*SectArray*) is *NSUPPORTS* long, with each array member representing a section-reference ID of a corresponding support.

SECTYPE (p. 1697) specifies the type of support, and **SECDATA (p. 1655)** specifies the geometry of the support.

Example 1: Specifying Support Information in an Additive Manufacturing Analysis

```
! specify supports
*dim,suppsect,,2               ! two supports
suppsect(1)=101                ! support 1 sectID=101
suppsect(2)=101                ! support 2 sectID=101
!
sectype,101,support,block      ! sectype is a block support
secdata,.07,1                  ! wall thickness and spacing
amsupport,2,support,suppsect   ! root name is "support" and suppsect is the
                               !   array name containing the section IDs
```

This command is also valid in PREP7.

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

AMTYPE, *Process*

Specifies the printing process in an additive manufacturing analysis.

SOLUTION (p. 38): Additive Manufacturing (p. 50)

Valid Products: (p. 2) Pro | Premium | Enterprise | PrepPost | Solver | **AS add-on**

Process

Process option:

PBF

Powder-bed fusion process.

DED

Directed-energy deposition process.

Command Default

None. A process option is required.

Notes

The powder-bed fusion (PBF) process uses thermal energy from a laser or electron beam to selectively fuse powder in a powder bed.

The directed-energy deposition (DED) process uses thermal energy, typically from a laser, to fuse materials by melting them as they are deposited.

This command is also valid in PREP7.

For more information, including a list of the elements and commands used in an additive manufacturing analysis, see Additive Simulation in Mechanical in the *LPBF Simulation Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/AN3D, *Keyword*, *KEY*

Specifies 3D annotation functions

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

If *Keyword* = ANUM, the type of annotation is defined, along with its location. The command format is **/AN3D (p. 129)**,ANUM,*NUM*,*TYPE*,*XHOT*,*YHOT*,*ZHOT*.

NUM

Unique number assigned as each annotation is applied to a model. These numbers are applied sequentially, although when an annotation entity is deleted, its number is reassigned.

TYPE

Annotation internal type number (101 = text, 102 = line, 103 = point, 104 = area, 105 = arrow, 106 = symbol, 108 = bitmap).

XHOT, YHOT, ZHOT

X, Y, Z coordinates for hot spot location.

If *Keyword* = BITM, the annotation is a bitmap. The command format is **/AN3D (p. 129)**,BITM,*TYPE*,*X*,*Y*,*Z*.

TYPE

An integer value between 1 and 99, indicating a texture or bitmap. Numbers 1 through 40 correspond to existing textures (see **/TXTRE (p. 2029)**). Numbers 51 through 99 correspond to textures defined using the File option of the **/TXTRE (p. 2029)** command. You can use this capability to override the predefined logo, clamp and arrow files available from the GUI dialog box (numbers 51 through 57). Numbers 41 through 50 are reserved.

X, Y, Z

X, Y, Z coordinates for the lower left corner of the bitmap.

If *Keyword* = TEXT, an annotation text string is created. The command format is **/AN3D (p. 129)**,TEXT,*X*,*Y*,*Z*,*Text_String*.

X, Y, Z

X, Y, Z coordinate location for text string.

Text_String

Text string to be applied as annotation.

If *Keyword* = LINE, an annotation line is created. The command format is **/AN3D (p. 129)**,LINE,*X1*,*Y1*,*Z1*,*X2*,*Y2*,*Z2*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X1, Y1, Z1

X, Y, Z coordinates for beginning of line.

X2, Y2, Z2

X, Y, Z coordinates for end of line.

If *Keyword* = POINT, a dot will be drawn in space (this option is not available from the GUI). The command format is **/AN3D (p. 129)**,POINT,*X*,*Y*,*Z*.

X, Y, Z

X, Y, Z coordinates for point.

If *Keyword* = AREA, a polygonal area with *n* vertices will be drawn. The command format is **/AN3D (p. 129)**,AREA,*NVERT*,*Xn*,*Yn*,*Zn*.

NVERT

The number of vertices (*n*) for the polygon. Your Polygon can have between 3 and 9 vertices.

Xn, Yn, Zn

X, Y, Z coordinate location for vertex *n*.

If *Keyword* = ARROW, an annotation arrow is created. The command format is **/AN3D (p. 129)**,ARROW,*SIZE*,*X1*,*Y1*,*Z1*,*X2*,*Y2*,*Z2*.

SIZE

Symbol size multiplier (0.1 to 20.0, default = 1.0) for the head of the arrow.

X1, Y1, Z1

X, Y, Z coordinates for the location of the tail.

X2, Y2, Z2

X, Y, Z coordinate for the location of the tip.

If *Keyword* = SYMBOL, an annotation symbol is created. The command format is **/AN3D (p. 129)**,SYMBOL,*TYPE*,*X*,*Y*,*Z*,*SIZE*.

TYPE

The symbol type (1 = CAP, 2 = TEE, 3 = CIRCLE, 4 = TRIANGLE, 5 = STAR).

X, Y, Z

X, Y, Z coordinate location for the symbol.

SIZE

Size multiplier for the symbol (0.1 to 20.0, default = 1.0)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Because 3D annotation is applied in relation to the XYZ coordinates of the anchor, you can transform your model, and the annotation will maintain the spatial relationship with the model. This works within reason, and there are instances where changing the perspective or the size of the model will change the apparent relationship between the annotation and the model.

The overall 3D dimensions of your model are defined by a bounding box. If portions of your model's bounding box lie outside of the visible area of your graphics window (if you are zoomed in on a specific area of your model), it can affect the placement of your 3D annotations. Zooming out will usually overcome this problem.

3D annotation is valid for the Cartesian (**CSYS (p. 441)**,0) coordinate system only. If you want to annotate a model you created in another coordinate system, use 2D annotation (note that 2D annotations do not remain anchored for dynamic rotations or transformations).

When you apply user defined bitmaps, the size of the annotation can vary. Use the options menu of the 3D annotation widget to adjust the size and placement of your bitmaps.

You cannot use the "!" and "$" characters in Mechanical APDL text annotation.

The GUI generates this command during 3D annotation operations and inserts the command into the log file (`Jobname.LOG`). You should NOT type this command directly during a Mechanical APDL session (although the command can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

Menu Paths

Utilty Menu>PlotCtrls>Annotate>Create 3D Annotation

ANCNTR, *NFRAM*, *DELAY*, *NCYCL*

Produces an animated sequence of a contoured deformed shape.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frames captures (defaults to 5).

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

ANCNTR (p. 131) involves a Mechanical APDL macro which produces an animation of a contoured deformed shape of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. After executing **ANCNTR (p. 131)**, you can replay the animated sequence by issuing the **ANIM (p. 142)** command.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Deformed Results

ANCUT, *NFRAM*, *DELAY*, *NCYCL*, *QOFF*, *KTOP*, *TOPOFF*, *NODE1*, *NODE2*, *NODE3*

Produces an animated sequence of Q-slices.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frames captures (defaults to 5).

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

QOFF

Q-slice working plane increment (defaults to .1 half screens).

KTOP

Topological effect on or off (YES or NO; default is NO).

TOPOFF

Topological offset (default is .1 half screens).

NODE1

Node 1 for start of the Q-slice.

NODE2

Node 2 for direction of the Q-slice.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NODE3

Node 3 for plane of the Q-slice.

Notes

ANCUT (p. 132) involves a Mechanical APDL macro which produces an animation of Q-slices of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. After executing **ANCUT (p. 132)**, you can replay the animated sequence by issuing the **ANIM (p. 142)** command.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Q-Slice Contours

Utility Menu>PlotCtrls>Animate>Q-Slice Vectors

ANCYC, *NUMFRAMES*, *KCYCL*, *DELAY*

Applies a traveling wave animation to graphics data in a modal cyclic symmetry analysis.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUMFRAMES

The number of plot frames for the animation. Valid values range from 5 through 36. The default is 18. A low value (because it specifies fewer graphical frames) produces a rougher animation but loads faster. A high value produces a smoother animation but requires more time to load.

KCYCL

The animation mode:

0

Continuous animation cycle (forward-reverse-forward).

1

Discontinuous animation cycle (forward-reset-forward). This option is the default.

DELAY

The time delay (in seconds) between animation frames. Valid values range from 0.1 through 1.0. The default is 0.1 seconds, which produces a seemingly real-time animation. A higher value produces a slower animation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

The default **ANCYC (p. 133)** command (issuing the command with no arguments) specifies these implicit argument values: **ANCYC (p. 133)**, 18, 0, 0.1

Notes

The **ANCYC (p. 133)** command is valid in a modal cyclic symmetry analysis only.

The command animates the cyclic symmetry mode shape plot in the General Post Processor (**/POST1 (p. 1461)**). When you issue a nodal- or element-results plot command (for example, **PLNSOL (p. 1425)**, **PLESOL (p. 1401)**, or **PLDISP (p. 1400)**) and then issue the **ANCYC (p. 133)** command, Mechanical APDL applies a traveling wave animation to the mode shape plot.

Each frame of the animation is created by expanding the cyclic symmetry mode shape at increasing phase angles (via the **/CYCEXPAND (p. 454)** command) starting at zero in equal increments over 360°. The phase-angle increment is 360 / *NUMFRAMES*.

The animation display shows the traveling wave of the result quantity being plotted. The traveling wave animation is applicable only to nodal diameters (harmonic indices) greater than 0 and less than *N* / 2 (where *N* is the number of cyclic sectors in the model).

For more information, see Applying a Traveling Wave Animation to the Cyclic Model in the *Cyclic Symmetry Analysis Guide*.

Menu Paths

Utility Menu> PlotCtrls> Animate> Cyc Traveling Wave

ANDATA, *DELAY*, *NCYCL*, *RSLTDAT*, *MIN*, *MAX*, *INCR*, *FRCLST*, *AUTOCONT*, *--*, *AUTOCNTR*

Displays animated graphics data for nonlinear problems.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DELAY

Time delay during animation (defaults to 0.5 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

RSLTDAT

The type of results data to be used for the animation sequence. This can be:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Current load step data (default).

1

Range of load step data.

2

Range of results data.

MIN

The range minimum value. If left blank or 0, defaults to the first data point.

MAX

The range maximum value. If left blank or 0, defaults to the last data point.

INCR

The increment between result data (defaults to 1).

FRCLST

Key to force the last sub step in a selected load step to be included in the animation (defaults to 0).

AUTOCONT

A value of 1 enables automatic scaling of contour values based on the overall subset range of values. The default value is 0 (no automatic scaling).

--

Unused field.

AUTOCNTR

A value of 1 disables automatic centering of displaced plots. The default value is 0 (allow automatic centering).

Notes

Use the **ANDATA (p. 134)** command to create animations for nonlinear problems. The command works by displaying an individual graphical image for each result data set from the results file. For information about creating animations for linear problems, see the **ANIM (p. 142)** command.

The command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. It uses a macro to produce an animation based on the last plot action command (for example, **PLDISP (p. 1400)**).

The results file must have more than one set of results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The command implicitly issues **/DSCALE (p. 575)**, 1 for default displacement scaling. Large displacements may not give good results.

This command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Animate Over Results

ANDSCL, *NFRAM*, *DELAY*, *NCYCL*

Produces an animated sequence of a deformed shape.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frames captured (defaults to 5).

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

Notes

ANDSCL (p. 136) involves a Mechanical APDL macro which produces an animation of displacement of the last plot action command (for example, **PLDISP (p. 1400)**). This command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. After executing **ANDSCL (p. 136)**, you can replay the animated sequence by issuing the **ANIM (p. 142)** command.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Deformed Shape

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANDYNA, *DELAY*, *NCYCL*, *START*, *END*, *INC*, *AUTOCONTOURKEY*

Produces an animated sequence of contour values through substeps.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

START

Number of the starting substep (defaults to 1).

END

Number of the ending substep (defaults to the maximum substep).

INC

Increment between substeps (defaults to 1).

AUTOCONTOURKEY

Auto-scales contour values, based on the overall subset range of values (defaults to 0, no auto-scaling).

Notes

ANDYNA (p. 137) involves a Mechanical APDL macro which produces an animation of contour values through all the substeps of the last plot action command. This command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. After executing **ANDYNA (p. 137)**, you can replay the animated sequence by issuing **ANIM (p. 142)**.

The command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Dynamic Results

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/ANFILE, *LAB*, *Fname*, *Ext*, - -

Saves or resumes an animation sequence to or from a file.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LAB

Label type.

SAVE

Save the current animation to a file.

RESUME

Resume an animation from a file.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to ANIM if *Fname* is blank.

- -

Unused field.

Notes

This command saves an animation to a file from local terminal segments or resumes an animation from a file to local terminal segments. See the **/SEG (p. 1707)** command for details on segment storage. See the **ANCNTR (p. 131)** macro for a convenient method of storing graphics frames in terminal memory segments. This command is device dependent and is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Restore Animation

Utility Menu>PlotCtrls>Animate>Save Animation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANFLOW, *NFRAM*, *DELAY*, *NCYCL*, *TIME*, *SPACING*, *SIZE*, *LENGTH*

Produces an animated sequence of a charged particle traveling in an electric or magnetic field.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frames captured (defaults to 5).

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Non-UI mode only.

TIME

Total Trace Time (seconds) (defaults to 0, which is the full flow trace).

SPACING

Particle spacing in seconds (defaults to 0).

SIZE

Particle size (defaults to 0, which is a line).

LENGTH

Particle length fraction (defaults to .1).

Notes

ANFLOW (p. 139) invokes a Mechanical APDL macro which produces an animation of charged particle motion in an electric or magnetic field by the last plot action command (that is, **PLTRAC (p. 1442)**). This command is only operational on graphic display platforms supporting the **/SEG (p. 1707)** command. After executing **ANFLOW (p. 139)**, you can replay the animated sequence by issuing the **ANIM (p. 142)** command. This command is functional only in the Postprocessor.

The *TIME* option lets you set the time interval of forward travel for the trace. The *SPACING* option is used to define the particle spacing in seconds from adjacent particles in the stream line. The *SIZE* variable sets the radius of the particle. The *LENGTH* variable is used to define the particle length fraction. By default, the *LENGTH* is set to .1, which means the particle occupies 10% of the trace region and the other 90% is a color-code line. The *SPACING* and *LENGTH* variables only make sense when the *SIZE* variable is nonzero (that is, the particle is bigger than the line).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Animate>Particle Flow

/ANGLE,*WN*,*THETA*,*Axis*,*KINCR*

Rotates the display about an axis.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

THETA

Angle (degrees) for changing display orientation (positive, counterclockwise about specified axis).

Axis

Rotation axis: XS, YS, or ZS (default) for the screen axes; XM, YM, or ZM for the global Cartesian model axes. ZS is normal to the screen; all axes pass through the focus point.

KINCR

Cumulative rotation key:

0

Do not use cumulative successive rotations.

1

Use cumulative rotations. Rotations are relative to the previous rotation. View settings (**/VIEW (p. 2131)**) are recalculated.

Notes

Default orientation is YS vertical. When the **/XFRM (p. 2194)** command is set for rotation about two points, or for entities, the **/ANGLE (p. 140)** command is functional only for *Axis* = ZS or ZM and *KINCR* = 1.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Angle of Rotation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANHARM, *NFRAM*, *DELAY*, *NCYCL*, *NPERIOD*, *CMS_ANTYPE*, *CMS_MODOPT*

Produces an animated sequence of time-harmonic results or complex mode shapes.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frame captures per cycle. Defaults to 12.

DELAY

Time delay (seconds) during animation. Defaults to 0.1 seconds.

NCYCL

Number of animation cycles. Defaults to 5. Not available in the GUI.

NPERIOD

Period number for the second set of frames showing the decay or growth of a mode shape. Only applies to complex mode shape animation. Defaults to 1, animating the decay or growth on the first period. Issue -1 to animate without decay or growth.

CMS_ANTYPE

Analysis type performed in the CMS use pass. No default.

MODAL

Modal analysis

HARMIC

Harmonic analysis

CMS_MODOPT

Mode extraction method selected in the CMS use pass. No default.

UNSYM

Unsymmetric matrix

DAMP

Damped system

QRDAMP

Damped system using QR algorithm

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

ANHARM (p. 141) invokes a Mechanical APDL macro which produces an animated sequence of:

- Time-harmonic results in the case of a harmonic analysis (**ANTYPE (p. 162)**,HARMIC)
- Complex mode shapes in the case of a modal analysis (**ANTYPE (p. 162)**,MODAL).

In both cases, the results are those of the last plot action (for example, **PLNSOL (p. 1425)**,B,SUM).

The animation converts the complex solution variables (real and imaginary sets) into time varying results over one period. For example, if *NFRAM* = 12, then the frame captures are in increments of 30 degree phase angles.

A second set of *NFRAM* frames will be generated for damped eigenmodes from complex eigensolvers to visualize any exponential decay or growth of the oscillations. The second set generated will display frames from the period number specified by *NPERIOD*.

In a CMS analysis, the **ANHARM (p. 141)** command can be used after the CMS expansion pass or the use pass. To use **ANHARM (p. 141)** after the expansion pass, you must indicate whether a modal analysis or a harmonic analysis was performed in the CMS use pass by setting *CMS_ANTYPE* to either MODAL or HARMIC. If the use pass was a modal analysis, you must also set the *CMS_MODOPT* field to indicate the mode extraction method that was used (UNSYM, DAMP, or QRDAMP). If *CMS_MODOPT* = QRDAMP, it is assumed that *CPXMOD* was set to ON in the **MODOPT (p. 1140)** command to request complex eigenmodes. If the **ANHARM (p. 141)** command is used after the use pass, it is not necessary to set the *CMS_ANTYPE* or *CMS_MODOPT* arguments.

For more information about complex results postprocessing, see POST1 and POST26 – Complex Results Postprocessing in the *Theory Reference*

Menu Paths

Utility Menu>PlotCtrls>Animate>Animate Over Time

Utility Menu>PlotCtrls>Animate>Time-harmonic

ANIM, *NCYCL*, *KCYCL*, *DELAY*

Displays animated graphics data for linear problems.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NCYCL

Number of cycles associated with the animation (defaults to 5 in non-GUI mode only)

KCYCL

Animation mode:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Continuous animation cycle (forward-reverse-forward-etc.) (default).

1

Discontinuous animation cycle (forward-reset-forward-etc.).

DELAY

Time delay (seconds) between animation frames (defaults to 0.1 seconds).

Notes

Use the **ANIM (p. 142)** command to create animations for linear problems only. The command uses the currently displayed picture based on one particular data set from the results file, and linearly interpolates that data into different sets, displaying pictures of each interpolated data set in sequence to create animation. For information about creating animations for nonlinear problems, see the **ANDATA (p. 134)** command.

This command is device-dependent.

Do not resize the graphic while animation is in progress; doing so can result in distorted plots.

For more information, see the **/SEG (p. 1707)** command for details about segment storage, and the **ANCNTR (p. 131)** macro for a convenient method of storing graphics frames in terminal memory segments.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Replay Animation

Utility Menu>PlotCtrls>Animate>Restore Animation

ANISOS, *NFRAM*, *DELAY*, *NCYCL*

Produces an animated sequence of an isosurface.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frames captures (defaults to 9).

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

ANISOS (p. 143) invokes a Mechanical APDL macro which produces an animation of an isosurface of the last plot action command (for example, **PLNSOL (p. 1425)**,S,EQV). The **ANISOS (p. 143)** command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. After executing **ANISOS (p. 143)**, you can replay the animated sequence by issuing the **ANIM (p. 142)** command.

This command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Isosurfaces

ANMODE, *NFRAM*, *DELAY*, *NCYCL*, *KACCEL*

Produces an animated sequence of a mode shape.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frames captures (defaults to 5).

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

KACCEL

Acceleration type:

0

Linear acceleration.

1

Sinusoidal acceleration.

Notes

ANMODE (p. 144) invokes a Mechanical APDL macro which produces an animation of mode shape of the last plot action command (for example, **PLDISP (p. 1400)**). The **ANMODE (p. 144)** command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. After executing **ANMODE (p. 144)**, you can replay the animated sequence by issuing the **ANIM (p. 142)** command.

This command functions only in the postprocessor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Animate>Mode Shape

/ANNOT, *Lab*, *VAL1*, *VAL2*

Activates graphics for annotating displays (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Annotation control key:

OFF

Turns off annotation for each subsequent display (default).

ON

Turns on annotation for each subsequent display.

DELE

Deletes all annotation.

SAVE

Saves annotation on a file. Use *VAL1* for file name (defaults to Jobname) and *VAL2* for the extension (defaults to ANO).

SCALE

Sets annotation scale factor (direct input only). Use *VAL1* for value (0.1 to 10.0) (defaults to 1.0).

XORIG

Sets the annotation x origin (direct input only). Use *VAL1* for value (-3.0 to 3.0).

YORIG

Sets annotation y origin (direct input only). Use *VAL1* for value (-3.0 to 3.0).

SNAP

Sets annotation snap (menu button input only). Use *VAL1* for value (0.002 to 0.2) (defaults to 0.002).

STAT

Displays current annotation status.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEFA

Sets annotation specifications to the default values.

REFR

Redisplays annotation graphics.

TMOD

Sets the annotation text mode. If *VAL1* = 1, annotation text will be drawn in scalable bitmap fonts (default). If *VAL1* = 0, annotation text will be drawn with stroke text.

VAL1

Value (or file name) as noted with label above.

VAL2

Value (or file name extension) as noted with label above.

Notes

This is a command generated by the GUI and will appear in the log file (`Jobname.log`) if annotation is used. This command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

You cannot use the "!" and "$" characters in Mechanical APDL text annotation.

/ANNOT (p. 145) activates annotation graphics for adding annotation to displays. Commands representing the annotation instructions are automatically created by the annotation functions in the GUI and written to `Jobname.log`.

The annotation commands are **/ANNOT (p. 145)**, **/ANUM (p. 166)**, **/TLABEL (p. 2007)**, **/LINE (p. 1051)**, **/LARC (p. 1011)**, **/LSYMBOL (p. 1088)**, **/POLYGON (p. 1460)**, **/PMORE (p. 1454)**, **/PCIRCLE (p. 1370)**, **/PWEDGE (p. 1557)**, **/TSPEC (p. 2024)**, **/LSPEC (p. 1081)**, and **/PSPEC (p. 1548)**. Annotation graphics are relative to the full Graphics Window and are not affected by Mechanical APDL window-specific commands (**/WINDOW (p. 2177)**, **/VIEW (p. 2131)**, etc.).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANORM,*ANUM*,*NOEFLIP*

Reorients area normals.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ANUM

Area number having the normal direction that the reoriented areas are to match.

NOEFLIP

Indicates whether you want to change the normal direction of the existing elements on the reoriented area(s) so that they are consistent with each area's new normal direction.

0

Make the normal direction of existing elements on the reoriented area(s) consistent with each area's new normal direction (default).

1

Do not change the normal direction of existing elements on the reoriented area(s).

Notes

Reorients areas so that their normals are consistent with that of a specified area.

If any of the areas have inner loops, the **ANORM (p. 147)** command will consider the inner loops when it reorients the area normals.

You cannot use the **ANORM (p. 147)** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

See Revising Your Model of the *Modeling and Meshing Guide* for more information.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Areas>Area Normals

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANPRES,*NFRAM*,*DELAY*,*NCYCL*,*RefFrame*

Produces an animated sequence of the time-harmonic pressure variation of an engine-order excitation in a cyclic harmonic analysis.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frame captures per cycle. Defaults to 3 times the number of sectors.

DELAY

Time delay (seconds) during animation. Defaults to 0.1 seconds.

NCYCL

Number of animation cycles. Defaults to 5.

RefFrame

Reference frame for the model rotation.

0

Rotating reference frame (default). The model remains fixed in space and the pressure revolve around the model.

1

Stationary reference frame. The model rotates and the pressure locations remain fixed in space.

Notes

ANPRES (p. 148) invokes a macro which produces an animated sequence of the time-harmonic applied pressure in the case of a mode-superposition harmonic analysis (**ANTYPE (p. 162)**,HARMIC with **CYCOPT (p. 466)**,MSUP,ON). The engine-order excitation must also have been specified (**CYCFREQ (p. 458)**,EO). While pressure loads are not accepted as valid loading in a mode-superposition analysis (they must be applied in the modal analysis and the modal load vector applied in the mode-superposition analysis) you can apply them for the purposes of this animation.

For *RefFrame* = 1 (stationary reference frame), the rotational velocity from the Linear Perturbation step, or the current **OMEGA (p. 1324)** or **CGOMGA (p. 320)** value, is used to determine the rotation direction about the cyclic cylindrical axis, otherwise a positive rotation is assumed.

You may use **/HBC (p. 887)**,,ON to hide overlapping pressure faces, and use **/GLINE (p. 860)**,,-1 to suppress the element outlines if desired.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSOL,*NVAR*,*NODE*,*Item*,*Comp*,*Name*,*Mat*,*Real*,*Ename*,*DataKey*

Specifies averaged element nodal data to be stored from the results file.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number assigned to this variable (2 to *NV* (**NUMVAR (p. 1302)**)). Overwrites any existing results for this variable.

NODE

Node number for which data are to be stored.

Item

Label identifying the item. General item labels are shown in Table 101: ANSOL - General Result Item and Component Labels (p. 151) below. Some items also require a component label. For selected result output, specify SRES and see Table 102: ANSOL - Selected Result (SRES) Component Labels (p. 153).

Comp

Component of the item (if required). General component labels are shown in Table 101: ANSOL - General Result Item and Component Labels (p. 151). Selected result components (*Item* = SRES) are shown in Table 102: ANSOL - Selected Result (SRES) Component Labels (p. 153).

Name

32-character name to identify the item on the printout and displays. Default: An eight-character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Mat

Material number. Average is calculated based on the subset of elements with the specified material number. Default: Use all elements in the active set unless *Real* and/or *Ename* is specified.

Real

Real number. Average is calculated based on the subset of elements with the specified real number. Default: Use all elements in the active set unless *Mat* and/or *Ename* is specified.

Ename

Element type name. Average is calculated based on the subset of elements with the specified element type name. Default: Use all elements in the active set unless *Mat* and/or *Real* is specified.

DataKey

Key to specify which data is stored:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AUTO

Nodal-averaged results are used if they are available for the first applicable time step; otherwise, the element-based data is used, if available. (Default.)

ESOL

Only element-based results are used. If they are not available, the command is ignored.

NAR

Only nodal-averaged results are used. If they are not available, the command is ignored.

Mat, *Real*, and *Ename* are ignored when nodal-averaged results are used.

Notes

Valid item and component labels for element nodal results are listed in Table 101: ANSOL - General Result Item and Component Labels (p. 151).

ANSOL (p. 149) defines element nodal results data to be stored from a results file (**FILE (p. 752)**). Not all items are valid for all nodes. See the input and output summary tables of each element attached to the node for the available items.

If nodal-averaged results (**OUTRES (p. 1336)**,NAR or another nodal-averaged label) are available, then **ANSOL (p. 149)** uses the nodal-averaged data for the applicable items (S, EPEL, EPPL, EPCR, EPTH) as dictated by the by *DataKey* argument. By default, (*DataKey* = AUTO), the availability of nodal-averaged results or element-based data is determined at the first load step that has results for the associated item. For more information, see Postprocessing Nodal-Averaged Results in the *Element Reference*.

Coordinate systems: Generally, element nodal quantities stored by **ANSOL (p. 149)** are obtained in the solution coordinate system (**RSYS (p. 1639)**, SOLU) and then averaged. There are some exceptions as listed below. **ANSOL (p. 149)** does not transform results from **RSYS (p. 1639)**,SOLU (or from the coordinate systems described for the exceptions below) to other coordinate systems. Verify that all elements attached to the subject node have the same coordinate system before using **ANSOL (p. 149)**.

- Layered element results are in the layer coordinate system (**RSYS (p. 1639)**,LSYS). You can further specify the element nodal results, for some elements, with the **SHELL (p. 1784)**, **LAYERP26 (p. 1016)**, and **FORCE (p. 772)** commands.
- When **ANSOL (p. 149)** is used to store nodal-averaged result data (based on the *DataType* setting), the global Cartesian coordinate system (**RSYS (p. 1639)**,0) is used.

Shell elements: The default shell element coordinate system is based on node ordering. For shell elements the adjacent elements could have a different **RSYS (p. 1639)**,SOLU, making the resultant averaged data inconsistent. A message to this effect is issued when **ANSOL (p. 149)** is used in models containing shell elements. Ensure that consistent coordinate systems are active for all associated elements used by the **ANSOL (p. 149)** command.

Derived quantities: Some of the result items supported by **ANSOL (p. 149)** (Table 101: ANSOL - General Result Item and Component Labels (p. 151)) are derived from the component quantities. Issue **AVPRIN (p. 210)** to specify the principal and vector sum quantity averaging methods.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default: If *Mat*, *Real* , and *Ename* are not specified, all elements attached to the node are considered. When a material ID, real constant ID, or element-type discontinuity is detected at a node, a message is issued. For example, in a FSI analysis, a FLUID30 element at the structure interface would be considered; however, because it contains no *SX* result, it is not used during **STORE (p. 1867)** operations.

Table 101: ANSOL - General Result Item and Component Labels

General Item and Component Labels ANSOL (p. 149), *NVAR, NODE, Item, Comp, Name, Mat, Real, Ename, DataType*		
Item	**Comp**	**Description**
S	X, Y, Z, XY, YZ, XZ	Component stress.[a]
	1, 2, 3	Principal stress.[a]
	INT	Stress intensity.[a]
	EQV	Equivalent stress.[a]
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.[a]
	1, 2, 3	Principal elastic strain.[a]
	INT	Elastic strain intensity.[a]
	EQV	Elastic equivalent strain.[a]
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.[a]
	1, 2, 3	Principal plastic strain.[a]
	INT	Plastic strain intensity.[a]
	EQV	Plastic equivalent strain.[a]
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.[a]
	1,2,3	Principal creep strain.[a]
	INT	Creep strain intensity.[a]
	EQV	Creep equivalent strain.[a]
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.[a]
	1, 2, 3	Principal thermal strain.[a]
	INT	Thermal strain intensity.[a]
	EQV	Thermal equivalent strain.[a]
ESIG	X, Y, Z, XY, YZ, XZ	Components of Biot’s effective stress.
	1, 2, 3	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
NL	SEPL	Equivalent stress (from stress-strain curve).
	SRAT	Stress state ratio.
	HPRES	Hydrostatic pressure.
	EPEQ	Accumulated equivalent plastic strain.
	CREQ	Accumulated equivalent creep strain.
	PSV	Plastic state variable.
	PLWK	Plastic work/volume.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ANSOL (p. 149), *NVAR,NODE,Item,Comp,Name,Mat,Real,Ename,DataType*		
Item	**Comp**	**Description**
CONT	STAT[b]	Contact status.
	PENE	Contact penetration.
	PRES	Contact pressure.
	SFRIC	Contact friction stress.
	STOT	Contact total stress (pressure plus friction).
	SLIDE	Contact sliding distance.
	GAP	Contact gap distance.
	FLUX	Total heat flux at contact surface.
	CNOS	Total number of contact status changes during substep.
	FPRS	Fluid penetration pressure.
TG	X, Y, Z, SUM[c]	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM[c]	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum.
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum.
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).
FFLX	X, Y, Z	Fluid-flow flux in poromechanics.
FGRA	X, Y, Z	Fluid pore-pressure gradient in poromechanics.
PMSV	VRAT, PPRE, DSAT, RPER	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure-thermal elements.
NS	X, Y, Z, XY, YZ, XZ	Nominal strain for hyperelastic material, reported in the current configuration (unaffected by **RSYS (p. 1639)**).
MPLA	DMAC, DMAX	Microplane damage, macroscopic and maximum values.
MPDP	TOTA, TENS, COMP, RW	Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).
EPFR		Free strain in porous media
DAMAGE	1,2,3,MAX	Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).
GDMG		Damage
IDIS		Structural-thermal dissipation rate

[a] This item stores nodal-averaged results if they are available on the results file.

[b] For more information about the meaning of contact status and its possible values, see Reviewing Results in POST1 in the *Contact Technology Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[c] *Comp* = SUM is not supported for coupled pore-pressure-thermal (CPT*nnn*) elements.

Table 102: ANSOL - Selected Result (SRES) Component Labels

ANSOL (p. 149),*NVAR*,*NODE*,SRES,*Comp*,*Name*,*Mat*,*Real*,*Ename*	
Comp	**Description**
SVAR*n*	The *n*th state variable.
FLDUF0*n*	The *n*th user-defined field variable.
ItemComp (concatenated *Item* + *Comp* label) from Table 208: OSRESULT - Item and Component Labels (p. 1328)[a]	See the table (p. 1328) for the combined item and component description.

[a] See Example 10: Specifying and Retrieving Selected Result Output (p. 1329).

Menu Paths

Main Menu>TimeHist Postpro>Variable Viewer

ANSTOAQWA,*Fname*,*VertAxis*,*Gc*,*Rho*,*HWL*,*DiffKey*,*SymxKey*,*SymyKey*

Creates an AQWA-LINE input file from the current Mechanical APDL model.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

AQWA file name. Defaults to Jobname.

VertAxis

Axis in the vertical direction:

Y (or 2)

Global Y axis.

Z (or 3)

Global Z axis (default).

Gc

Gravitational acceleration. Defaults to 9.81.

Rho

Density of water. Defaults to 1025.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HWL

Waterline height in model coordinates. Defaults to 0.0.

DiffKey

Diffracting model key:

0

Create a non-diffracting AQWA model.

1

Create a diffracting AQWA model (default).

SymxKey

Key indicating if model is symmetric about the global XZ plane:

0

No symmetry about XZ plane (default).

1

Use symmetry about XZ plane. Only include (or select) half the model.

SymyKey

Key indicating if model is symmetric about the global YZ plane:

0

No symmetry about YZ plane (default).

1

Use symmetry about YZ plane. Only include (or select) half the model.

Notes

This command creates the input file *Fname*.aqwa for the Ansys Aqwa Multi-Body Hydrodynamics System for diffraction analysis in AQWA-LINE from the model currently in the database, based on the currently selected set of elements. The selected set must only include the hull envelope; no internal structure should be selected.

There should be a line of nodes defined at the waterline. Only those elements that are entirely below the waterline will be specified as diffracting. If there are no waterline nodes, there will be no diffracting elements at the waterline, which will severely reduce the accuracy of the diffraction analysis.

The translator maps PLANE42, SHELL63, and SHELL181 elements to PANELs, and maps PIPE16 and PIPE59 elements to TUBEs. It does not recognize any other element types. Any material or geometric properties

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can be used for the shell elements, as AQWA does not need any properties at all and the command does not use them. All the shell elements below the water must have their normals pointing outward.

TUBE elements in AQWA have material density, outside diameter, wall thickness, added mass, and drag coefficients, so appropriate properties should be used in the Mechanical APDL model. PIPE59 elements can have added mass and damping coefficients; these will be written to the file. The Mechanical APDL program uses the inertia coefficient C_M, whereas AQWA uses the added mass coefficient C_A, where $C_M = (1 + C_A)$. This correction is made automatically.

Documentation for archived elements PIPE16, PLANE42, PIPE59, and SHELL63 appears in the *Feature Archive*.

In AQWA the vertical axis is always the Z-axis. The command can convert a model built with either the Y or Z-axis vertical, but the X-axis must be horizontal and should preferably be along the fore/aft axis of the vessel. If the structure is symmetric and you wish to use the symmetry options, you must only select one half or one quarter of the model, as appropriate. If you model a complete vessel and specify X symmetry, the AQWA model will contain two sets of coincident elements.

If you are working from a model created for a structural analysis, it will probably be necessary to remesh the model as the structural mesh is most likely finer than needed for a diffraction analysis.

If you enter this command interactively (with the GUI active) and no data is provided for the command options, the application prompts you for their values.

You must verify the completeness and accuracy of the data written.

AQWA-LINE Notes

The file will specify restart stages 1-2 only. It has no options except REST, so AQWA may fail if any of the elements are badly shaped.

The total mass is obtained by integrating over the wetted surface area and adding the TUBE masses, so it should be reasonably accurate. However, the integration used is not as accurate as that in AQWA, so there may be a small difference between the weight and buoyancy, particularly if tubes represent a large portion of the model.

The position of the CG is unknown. A point mass is placed at the water-line above the CB, but you should change this to the correct position.

The moments of inertia are estimated based on the overall dimensions of the model and using standard formulae for a ship. You should change these to the correct values.

The maximum frequency is calculated from the maximum side length of the underwater elements. The range of frequencies runs from 0.1 rad/s to the calculated maximum, in steps of 0.1 rad/s.

The directions are in steps of 15° over a range that is determined by the symmetry you have specified, in accordance with the requirements of AQWA.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANSTOASAS, *Fname*, *KEY*

Creates an ASAS input file from the current Mechanical APDL model.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

ASAS file name. Defaults to Jobname.

KEY

Key indicating type of file to produce:

0

ASAS file for use by Ansys Aqwa (no loads written). Creates the file *Fname*.asas.

1

ASAS file (all data written, including loads). Creates the file *Fname*.asas.

2

ASAS(NL) file. Creates the file *Fname*.asnl.

Notes

This command creates an input file for the Ansys Asas Finite Element Analysis System from the model and loads currently in the database, based on the currently selected set of elements. Most common structural element types are written, as well as sections (or real constants), materials, boundary conditions and loads, and solution and load step options.

Data Written

The following data is written:

- Solution control options (p. 157)
- Nodes
- Elements (p. 157)
- Material data (p. 159)
- Geometry data (p. 159)
- Section data (p. 160)
- Mechanical APDL element components (ASAS sets)
- Boundary conditions (p. 160)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Loads (p. 160)
- Added mass (via MASS21 element)

Details are provided in the following sections.

Not all data is written. You must verify the completeness and accuracy of the data. Only loading at the current step is transferred; hence, no load step history is captured.

Solution Control Options

The ASAS project name is defined as "Ansys".

The solution control options are converted as follows:

Mechanical APDL Analysis Type	ASAS(L) / AQWA-WAVE Option	ASAS(NL) Option
Static (0)	JOB NEW LINE	JOB STAT
Buckling (1)	Not applicable	JOB STAT SPIT KGEOM
Modal (2)	JOB NEW FREQ	JOB STAT SPIT KGEOM
Transient (4)	Not applicable	JOB TRAN
Remaining analysis types	JOB NEW LINE	JOB STAT

For conversion to ASAS(NL), the large displacement option is set based on **NLGEOM (p. 1231)**, final load solution time is set based on **TIME (p. 2000)**, and sub-step times are set based on **DELTIM (p. 515)** or **NSUBST (p. 1287)** (assuming constant step size).

Element Data

If you intend to use the data only with AQWA-WAVE, only the elements that form the wetted surface are required. Selecting these elements before invoking the **ANSTOASAS (p. 156)** command will improve performance. In order for AQWA-WAVE to identify the direction of the wave loading, all elements must be defined by nodes in a clockwise direction. For further information, refer to the AQWA-WAVE manual.

The element types are converted as follows:

Element Type	Supported Facilities	Notes
COMBIN14 - Spring-Damper	SPR1 SPR2 if rotational spring FLA2 (ASAS(L) only) if nodes are not coincident and longitudinal spring	ASAS(L) does not support spring elements with non-coincident nodes. In this case, COMBIN14 is converted to FLA2.
MASS21 - Structural Mass	N/A	In ASAS, additional mass is not added as an element. Hence, if this element is included, equivalent ASAS

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Type	Supported Facilities	Notes
		lumped added mass information is written.
PLANE42 - 2D Structural Solid [1]	QUM4 TRM3 - if Triangular	-
SOLID45 - 3D Structural Solid [1]	BRK8 TET4 - if Tetrahedral BRK6 - if Prism	TET4 elements are only available in ASAS(L). Element is not converted if Pyramidal.
SHELL61 - Axisymmetric-Harmonic Structural Shell	ASH2	ASH2 elements are only available in ASAS(L).
SHELL63 - Elastic Shell [1]	QUS4 TBC3 - if Triangular	TBC3 elements are only available in ASAS(L).
PLANE82 - 2D 8-Node Structural Solid [1]	QUM8 TRM6 - if Triangular	-
SOLID92 - 3D 10-Node Tetrahedral Structural Solid [1]	TE10	-
SOLID95 - 3D 20-Node Structural Solid [1]	BR20 TE10 - if Tetrahedral BR15 - if Prism	TE10 elements are only available in ASAS(L). Element is not converted if Pyramidal.
LINK180 - 3D Finite Strain Spar (or Truss)	FLA2	-
SHELL181 - 4-Node Finite Strain Shell	QUS4 TBC3 - if Triangular	TBC3 elements are only available in ASAS(L).
PLANE182 - 2D 4-Node Structural Solid	QUM4 TRM3 - if Triangular	-
PLANE183 - 2D 8-Node or 6-Node Structural Solid	QUM8 TRM6 - if Triangular	-
SOLID185 - 3D 8-Node Structural Solid or Layered Solid	BRK8 TET4 - if Tetrahedral BRK6 - if Prism	TET4 elements are only available in ASAS(L). Element is not converted if Pyramidal.
SOLID186 - 3D 20-Node Structural Solid or Layered Solid	BR20 TE10 - if Tetrahedral BR15 - if Prism	TE10 elements are only available in ASAS(L). Element is not converted if Pyramidal.
SOLID187 - 3D 10-Node Tetrahedral Structural Solid	TE10	TE10 elements are only available in ASAS(L).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Type	Supported Facilities	Notes
BEAM188 - 3D Linear Finite Strain Beam	BM3D	-
BEAM189 - 3D Quadratic Finite Strain Beam	TCBM - if ASAS(L) STF4 - if ASAS(NL)	Refer to geometry details for limitations for TCBM elements. An orientation node is needed for STF4 and must be specified.
SHELL208 - 2-Node Finite Strain Axisymmetric Shell	ASH2	ASH2 elements are only available in ASAS(L).
SHELL281 - 8-Node Finite Strain Shell	TCS8	-
PIPE288 - 3D Linear Finite Strain Pipe	TUBE	-

1. Documentation for this archived element type appears in the *Feature Archive*.

Material Data

Linear isotropic material conversion is supported for ASAS and ASAS(NL).

Geometry Data

The following ASAS element geometry data is supported:

ASAS Element Type	Support Facilities	Notes
BM3D	Sections, orientation by 3rd node position.	Sections are always defined separately.
TUBE	Thickness and diameter defined, orientation by 3rd node or default local axes.	Using the default local axes will result in BETA being set to 90° (to ensure that the ASAS local axes are the same as those in Mechanical APDL).
BM2D	A, IZ & AY properties defined.	
TCBM	Uniform section properties supported.	TCBM does not support general local axis orientations. Hence, elements will only be correct if they lie in the global XY plane.
STF4	CTUB, RECT, and HREC Mechanical APDL subtypes supported.	STF4 elements are only supported in ASAS(NL). Local y and z are 90° to the Mechanical APDL definition.
FLA2	Uniform cross sectional area.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASAS Element Type	Support Facilities	Notes
SPR1 / SPR2	Stiffness (and also linear damping if ASAS(NL)) included.	
All non-beam elements	Constant element thickness.	

For all beam elements, the third node position must be explicitly defined. If the position is not defined, the program generates an error code (-1) in the output file.

Section Data

No user sections are generated if AQWA-WAVE data is selected.

The following sections are converted for ASAS and ASAS(NL):

Mechanical APDL Section Type	ASAS Section Type	Notes
CTUB	TUB	Tubular section
I	FBI	Fabricated I beam
HREC	BOX	Fabricated box
All others	PRI	Prismatic section, only flexural properties defined.

Boundary Conditions

The following boundary conditions are converted for ASAS and ASAS(NL):

Mechanical APDL Boundary Condition	ASAS Boundary Condition	Notes
Nodal U* and ROT* constraints	SUPPressed freedoms: X, Y, Z, RX, RY, RZ	Skewed systems are not supported.
Nodal U* and ROT* imposed non-zero values	DISPlaced freedoms: X, Y, Z, RX, RY, RZ	Skewed systems are not supported.
CP (p. 421) and **CE (p. 297)** constraint equations	CONStraint equation data	Skewed systems are not supported.

Loads

No user loading is generated if AQWA-WAVE data is selected. However, a load case (number 1000) is automatically defined to identify the wetted surface of the elements for use by AQWA-WAVE based on the normal surface loads applied to the solid or shell elements.

Pressure loads from SURF154 elements are converted to equivalent nodal loads for ASAS. For AQWA-WAVE, the SURF154 pressures are used to identify the wetted surface of the underlying elements. The following loads are converted for ASAS:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mechanical APDL Load Type	ASAS Load Type	Notes
SFE (p. 1761) (PRES)	PRESSURE (no sub-types)	Element families supported: Solids, shells (excluding edge pressures), planes (edge pressures only), and axisymmetric shells.
SFE (p. 1761) (PRES)	DISTRIBUted loading Shells - ML2 Beams - BL1 / BL2 Tubes - GL1 Curved beams - CB1	Element families supported: Shells (edge pressures only), Beams (includes tubes and curved beams).
F (p. 729) (F* and M*)	NODAL Load, in X, Y, Z, RX, RY, RZ	Skewed systems are not supported.
D (p. 483) (U* and ROT*)	PRESCRIBed displacements	
ACEL (p. 92)	BODY FORce	

Menu Paths

This command cannot be accessed from a menu.

ANTIME, *NFRAM*, *DELAY*, *NCYCL*, *AUTOCNTRKY*, *RSLTDAT*, *MIN*, *MAX*

Generates a sequential contour animation over a range of time.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFRAM

Number of frame captures (defaults to 5).

DELAY

Time delay during animation (defaults to 0.1 seconds).

NCYCL

Number of animation cycles (defaults to 5). Available in non-UI mode only.

AUTOCNTRKY

Auto-scales contour values, based on the overall subset range of values. The auto-scaling option defaults to 0, no auto-scaling.

RSLTDAT

The results data to be used for the animation sequence. This can be:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Current load step data (default).

1

Range of load step data.

2

Range of time data.

MIN

The range minimum value. If left blank defaults to the first data point.

MAX

The range maximum value. If left blank defaults to the last data point.

Notes

The **ANTIME (p. 161)** command operates only on graphic display platforms supporting the **/SEG (p. 1707)** command. It uses a Mechanical APDL macro to produce an animation of contour values for the last plot action command (for example, **PLDISP (p. 1400)**). After executing **ANTIME (p. 161)**, the **ANIM (p. 142)** command replays the animated sequence.

This command functions only in the postprocessor.

Menu Paths

Utility Menu>PlotCtrls>Animate>Animate Over Time

Utility Menu>PlotCtrls>Animate>Time-harmonic

ANTYPE,*Antype*,*Status*,*LDSTEP*,*SUBSTEP*,*Action*,--,*PRELP*

Specifies the analysis type and restart status.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Product Restrictions (p. 165)

Antype

Analysis type (defaults to the previously specified analysis type, or to STATIC if none specified):

STATIC or 0

Perform a static analysis. Valid for all degrees of freedom.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BUCKLE or 1

Perform a buckling analysis. Implies that a previous static solution was performed with prestress effects calculated (**PSTRES (p. 1551)**,ON). Valid for structural degrees of freedom only.

MODAL or 2

Perform a modal analysis. Valid for structural and fluid degrees of freedom.

HARMIC or 3

Perform a harmonic analysis. Valid for structural, fluid, magnetic, and electrical degrees of freedom.

TRANS or 4

Perform a transient analysis. Valid for all degrees of freedom.

SUBSTR or 7

Perform a substructure analysis. Valid for all degrees of freedom.

SPECTR or 8

Perform a spectrum analysis. Implies that a previous modal analysis was performed. Valid for structural degrees of freedom only.

SOIL or 9

Perform a soil analysis including geostatic stress equilibrium or consolidation. Valid for structural and fluid-pore-pressure degrees of freedom.

Status

Specifies the status of the analysis (new or restart):

NEW

Specifies a new analysis (default). If NEW, the remaining fields on this command are ignored.

RESTART

Specifies a restart of a previous analysis. Valid for static, modal, and transient (full or mode-superposition method) analyses. For more information about restarting static and transient analyses, see Multiframe Restart in the *Basic Analysis Guide*. For more information on restarting a modal analysis, see Modal Analysis Restart in the *Basic Analysis Guide*.

Multiframe restart is also valid for harmonic analysis, but is limited to 2D magnetic analysis only.

A substructure analysis (backsubstitution method only) can be restarted for the purpose of generating additional load vectors. For more information, see the **SEOPT (p. 1715)** command and Applying Loads and Creating the Superelement Matrices in the *Substructuring Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LDSTEP

Specifies the load step at which a multiframe restart begins.

For full transient and nonlinear static analyses, the default is the highest load step number found in the Jobname.R*nnn* files for the current jobname in the current directory.

For mode-superposition transient analyses, the default is none.

SUBSTEP

Specifies the substep at which a multiframe restart begins.

For full transient and nonlinear static analyses, the default is the highest substep number found for the specified *LDSTEP* in the Jobname.R*nnn* files in the current directory.

For mode-superposition transient analyses, the default is none.

Action

Specifies the manner of a multiframe restart.

CONTINUE

The program continues the analysis based on the specified *LDSTEP* and *SUBSTEP* (default). The current load step is continued. If the end of the load step is encountered in the .R*nnn* file, a new load step is started. The program deletes all .R*nnn* files, or .M*nnn* files for mode-superposition transient analyses, beyond the point of restart and updates the .ldhi file if a new load step is encountered.

ENDSTEP

At restart, force the specified load step (*LDSTEP*) to end at the specified substep (*SUBSTEP*), even though the end of the current load step has not been reached. At the end of the specified substep, all loadings are scaled to the level of the current ending and stored in the .ldhi file. A run following this ENDSTEP starts a new load step. This capability allows you to change the load level in the middle of a load step. The program updates the .ldhi file and deletes all .R*nnn* files, or .M*nnn* files for mode-superposition transient analyses, beyond the point of ENDSTEP. The .R*nnn* or .M*nnn* file at the point of ENDSTEP are re-written to record the rescaled load level.

RSTCREATE

At restart, retrieve information to be written to the results file for the specified load step (*LDSTEP*) and substep (*SUBSTEP*). Be sure to use **OUTRES (p. 1336)** to write the results to the results file. This action does not affect the .ldhi or .R*nnn* files. Previous items stored in the results file at and beyond the point of RSTCREATE are deleted. This option cannot be used to restart a mode-superposition transient analysis.

PERTURB

At restart, a linear perturbation analysis (static, modal, buckling, or full harmonic) is performed for the specified load step (*LDSTEP*) and substep (*SUBSTEP*). This action does not affect the .ldhi, .R*nnn*, or .RST files.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a linear perturbation analysis, set *Action* = PERTURB; otherwise, the existing restart files, such as the .ldhi, .R*nnn*, or .RST file, may be modified by the linear perturbation analysis. Issue the **PERTURB (p. 1378)** command to indicate the desired analysis type (STATIC, MODAL, BUCKLE, HARMONIC, or SUBSTR).

- -

Unused field.

PRELP

Flag indicating whether a subsequent linear perturbation will be performed:

YES

Specifies the first static analysis with a sequential linear perturbation analysis. Setting PRELP = YES is necessary for acoustics-structural interaction linear perturbation analysis, if the **MORPH (p. 1156)** command with *StrOpt* = YES is not issued.

NO

No specification for a subsequent linear perturbation (default).

Command Default

New static analysis.

Notes

If using the **ANTYPE (p. 162)** command to change the analysis type in the same SOLVE session, the program issues the following message: "Some analysis options have been reset to their defaults. Please verify current settings or respecify as required." Typically, the program resets commands such as **NLGEOM (p. 1231)** and **EQSLV (p. 648)** to their default values.

To read in view factors after restarting a radiation analysis, issue **VFOPT (p. 2099)**,READ after **ANTYPE (p. 162)**,,RESTART.

The analysis type (*Antype*) cannot be changed if a restart is specified. Always save parameters before doing a restart. For more information on the different types of restart, see Restarting an Analysis in the *Basic Analysis Guide*.

Issue **/CLEAR (p. 350)**,NOSTART *before* performing a restart of a previous nonlinear mesh adaptivity via **ANTYPE (p. 162)**,,RESTART,,,CONTINUE or **ANTYPE (p. 162)**,,RESTART,,,ENDSTEP.

This command is also valid in PREP7.

Product Restrictions

Command Option Antype	Available Products
STATIC	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BUCKLE	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MODAL	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
HARMONIC	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
TRANS	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SUBSTR	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SPECTR	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>New Analysis

Main Menu>Preprocessor>Loads>Analysis Type>Restart

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Analysis Type>New Analysis

Main Menu>Solution>Analysis Type>Restart

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

/ANUM, *NUM*, *TYPE*, *XHOT*, *YHOT*

Specifies the annotation number, type, and hot spot (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUM

Annotation number. Mechanical APDL automatically assigns the lowest available number. You cannot assign a higher number if a lower number is available; the program substitutes the lowest available number in place of any user-specified higher number.

TYPE

Annotation internal type number. If *TYPE* = DELE, delete annotation *NUM*.

1

Text

2

Block text (not available in GUI)

3

Dimensions

4

Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5

Rectangles

6

Circles

7

Polygons

8

Arcs

9

Wedges, pies

11

Symbols

12

Arrows

13

Bitmap

XHOT

X hot spot (-1.0 < X < 2.0). Used for menu button item delete.

YHOT

Y hot spot (-1.0 < Y < 1.0). Used for menu button item delete.

Command Default

Number, type, and hot spot are automatically determined.

Notes

This is a command generated by the GUI and will appear in the log file (`Jobname.log`) if annotation is used. This command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

Type 13 (bitmap) annotation applies user defined bitmaps defined using the FILE option of the **/TXTRE (p. 2029)** command.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

AOFFST, *NAREA*, *DIST*, *KINC*

Generates an area, offset from a given area.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NAREA

Area from which generated area is to be offset. If *NAREA* = ALL, offset from all selected areas (**ASEL (p. 192)**). If *NAREA* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

DIST

Distance normal to given area at which keypoints for generated area are to be located. Positive normal is determined from the right-hand-rule keypoint order.

KINC

Keypoint increment between areas. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

Notes

Generates an area (and its corresponding keypoints and lines) offset from a given area. The direction of the offset varies with the given area normal. End slopes of the generated lines remain the same as those of the given pattern. Area and line numbers are automatically assigned, beginning with the lowest available values (**NUMSTR (p. 1301)**).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Offset

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AOVLAP, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Overlaps areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA9

Numbers of areas to be operated on. If *NA1* = ALL, use all selected areas and ignore *NA2* to *NA9*. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1*.

Notes

Generates new areas which encompass the geometry of all the input areas. The new areas are defined by the regions of intersection of the input areas, and by the complementary (non-intersecting) regions. See Solid Modeling in the *Modeling and Meshing Guide* for an illustration. This operation is only valid when the region of intersection is an area. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Areas

APLOT, *NA1*, *NA2*, *NINC*, *DEGEN*, *SCALE*

Displays the selected areas.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NINC

Displays areas from *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas (**ASEL (p. 192)**) are displayed.

DEGEN

Degeneracy marker:

(blank)

No degeneracy marker is used (default).

DEGE

A red star is placed on keypoints at degeneracies (see the *Modeling and Meshing Guide*). Not available if **/FACET (p. 731)**,WIRE is set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SCALE

Scale factor for the size of the degeneracy-marker star. The scale is the size in window space (-1 to 1 in both directions) (defaults to .075).

Notes

This command is valid in any processor. The degree of tessellation used to plot the selected areas is set through the **/FACET (p. 731)** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>Plot Degen Areas

Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>Plot Degen Areas

Utility Menu>Plot>Areas

Utility Menu>Plot>Specified Entities>Areas

APORT, *PortNum*, *Label*, *KCN*, *PRES*, *PHASE*, --, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Specifies input data for plane wave and acoustic duct ports.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

PortNum

Port number. This number is associated with an exterior port or interior port previously specified by the **SF (p. 1733)** and **BF (p. 221)** family of commands, respectively. The number must be between 1 and 50.

Label

PLAN

Incident plane wave.

RECT

Rectangular duct.

CIRC

Circular duct.

COAX

Coaxial duct.

LIST

List the port settings. If *PortNum* = ALL, list the port settings for all defined ports.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELE

Delete defined ports. If *PortNum* = ALL, delete all defined ports.

KCN

A previously-defined local (*KCN* >10) or global (*KCN* = 0) Cartesian coordinate system number used to specify the geometric properties of the duct. Defaults to the global Cartesian coordinate system (0). The local Z-direction must be the direction of wave propagation. The origin of the local coordinate system must be centered about the face of the duct port without considering symmetry.

PRES

Zero-to-peak amplitude of the pressure. If blank, the port will appear as a matching impedance.

PHASE

Phase angle of the applied pressure in degrees. Defaults to 0.

--

Unused field.

VAL1, VAL2, VAL3, VAL4

Additional input. The meaning of *VAL1* through *VAL4* varies depending on the specified *Label*.

Label = PLAN:

VAL1

ϕ angle from positive X-axis to positive Y-axis in the local Cartesian coordinates (*KCN*).

VAL2

θ angle away from positive Z-axis in the local Cartesian coordinates (*KCN*).

VAL3 - VAL4

Not used.

Label = RECT:

VAL1

Width of the rectangular duct.

VAL2

Height of the rectangular duct.

VAL3

Mode index for pressure variation along the width (defaults to 0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL4

Mode index for pressure variation along the height (defaults to 0).

Label = CIRC:

VAL1

Radius of the circular duct.

VAL2

Not used.

VAL3

Mode index for pressure variation along the azimuth (defaults to 0).

VAL4

Mode index for pressure variation along the radii (defaults to 0).

Label = COAX:

VAL1

Inner radius of the coaxial duct.

VAL2

Outer radius of the coaxial duct.

VAL3

Mode index for pressure variation along the azimuth (defaults to 0).

VAL4

Mode index for pressure variation along the radii (defaults to 0).

Notes

Use the **APORT (p. 170)** command to launch a specified analytic acoustic mode into a guided duct.

The low-order FLUID30 element does not support the higher modes in the coaxial duct (*Label* = COAX).

For more information, see Specified Mode Excitation in an Acoustic Duct in the *Acoustic Analysis Guide*, and Analytic Port Modes in a Duct in the *Mechanical APDL Theory Reference*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

APPEND, *LSTEP*, *SBSTEP*, *FACT*, *KIMG*, *TIME*, *ANGLE*, *NSET*

Reads data from the results file and appends it to the database.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSTEP

Load step number of the data set to be read. Defaults to 1. If FIRST, ignore *SBSTEP* and *TIME* and read the first data set. If LAST, ignore *SBSTEP* and *TIME* and read the last data set. If NEXT, ignore *SBSTEP* and *TIME* and read the next data set. If already at the last data set, the next set is the first data set. If NEAR, ignore *SBSTEP* and read the data set nearest to *TIME*. If *TIME* is blank, read the first data set. If LIST, scan the results file to produce a summary of each load step (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored).

SBSTEP

Substep number (within *LSTEP*) (defaults to last substep of load step). For the Buckling (**ANTYPE (p. 162)**,BUCKLE) or Modal (**ANTYPE (p. 162)**,MODAL) analysis, the substep corresponds to the mode number (defaults to first mode). If *LSTEP* = LIST, *SBSTEP* = 0 or 1 will list the basic load step information; *SBSTEP* = 2 will also list the load step title, and label the imaginary data sets if they exist.

FACT

Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. Harmonic velocities or accelerations may be calculated from the displacement results from a modal or harmonic (**ANTYPE (p. 162)**,HARMIC) analyses. If *FACT* = VELO, the harmonic velocities (v) are calculated from the displacements (d) at a particular frequency (f) according to the relationship $v = 2 \pi f d$. Similarly, if *FACT* = ACEL, the harmonic accelerations (a) are calculated as $a = (2 \pi f)^2 d$.

KIMG

Used only with results from complex analyses:

0

Store real part of complex solution.

1

Store imaginary part.

TIME

Time-point identifying the data set to be read. For harmonic analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *LSTEP* is NEAR, read the data set nearest to *TIME*. If both *LSTEP* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

file (**OUTRES (p. 1336)**) for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, the last time point is used.

ANGLE

Circumferential location (0° to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (PLANE25, PLANE75, PLANE78, PLANE83, and SHELL61) of the load case. See the *Mechanical APDL Theory Reference* for details. Note that factored values of applied constraints and loads will overwrite any values existing in the database.

NSET

Data set number of the data set to be read. If a positive value for *NSET* is entered, *LSTEP*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **APPEND (p. 173)**,LIST. To determine if data sets are real or imaginary, issue **APPEND (p. 173)**,LIST,2 which labels imaginary data sets.

Notes

Reads a data set from the results file and appends it to the existing data in the database for the selected model only. The existing database is not cleared (or overwritten in total), allowing the requested results data to be merged into the database. Various operations may also be performed during the read operation. The database must have the model geometry available (or used the **RESUME (p. 1601)** command before the **APPEND (p. 173)** command to restore the geometry from `File.db`).

Menu Paths

Main Menu>General Postproc>Read Results>By Load Step

Main Menu>General Postproc>Read Results>By Set Number

Main Menu>General Postproc>Read Results>By Time/Freq

APTN, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NA7*, *NA8*, *NA9*

Partitions areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA9

Numbers of areas to be operated on. If *NA1* = ALL, *NA2* to *NA9* are ignored and all selected areas are used. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may be substituted for *NA1*.

Notes

Partitions areas that intersect. This command is similar to the combined functionality of the **ASBA (p. 186)** and **AOVLAP (p. 169)** commands. If the intersection of two or more areas is an area (that is, planar),

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

new areas will be created with boundaries that conform to the area of intersection and to the boundaries of the non-intersecting portions of the input areas (**AOVLAP (p. 169)**). If the intersection is a line (that is, not planar), the areas will be subtracted, or divided, along the line(s) of intersection (**ASBA (p. 186)**). Both types of intersection can occur during a single **APTN (p. 174)** operation. Areas that do not intersect will not be modified. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Areas

---

ARCLEN, *Key*, *MAXARC*, *MINARC*

Activates the arc-length method.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Arc-length key:

OFF

Do not use the arc-length method (default).

ON

Use the arc-length method.

MAXARC

Maximum multiplier of the reference arc-length radius (default = 25).

MINARC

Minimum multiplier of the reference arc-length radius (default = 1/1000).

Notes

Activates the arc-length method and sets the minimum and maximum multipliers for controlling the arc-length radius based on the initial arc-length radius.

The initial arc-length radius, t_0, is proportional (in absolute value) to the initial load factor. The initial load factor is given by:

Initial Load Factor = *TIME* / *NSBSTP*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where *TIME* is the time specified by the **TIME (p. 2000)** command for the arc-length load step, and *NSBSTP* is the number of substeps specified by the **NSUBST (p. 1287)** command.

The factors *MAXARC* and *MINARC* are used to define the range for the arc-length radius to expand and shrink during the substep solution:

t_{MAX} = *MAXARC* * t_0

t_{MIN} = *MINARC* * t_0

In each substep, the arc-length radius is kept constant throughout the equilibrium iterations. After each converged substep, the arc-length radius for the next substep is modified depending on the convergence behavior. If the substep converges and the program heuristic predicts an easy convergence, the arc-length radius is enlarged. If the enlarged value is greater than t_{MAX}, the arc-length radius is reset to t_{MAX}. If the substep does not converge, bisection will take place until the arc-length radius is reduced to t_{MIN}. If further nonconvergence is encountered, the solution terminates.

The arc-length method predicts the next time increment (that is, load factor increment). Therefore, the **AUTOTS (p. 207)** and **PRED (p. 1476)** commands are ignored when the arc-length method is used.

The **STABILIZE (p. 1858)** and **LNSRCH (p. 1057)** commands are also ignored.

The arc-length method cannot be used in a multiframe restart.

For difficult problems, one suggestion is to increase the initial number of substeps (**NSUBST (p. 1287)**), and to prevent the arc-length radius from increasing too rapidly (*MAXARC* = 1).

ARCLEN (p. 175) cannot be used for any load step that has no applied load or displacement.

The arc-length method does not support tabular loads. In order to use the arc-length method, you must replace tabular loads by other load types and then run the analysis again.

The arc-length method can only be used with the sparse solver (**EQSLV (p. 648)**,SPARSE). If any other solver is specified, the solver method is automatically changed to sparse, and a warning message is issued to notify you.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Arc-Length Opts

Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL

Main Menu>Solution>Load Step Opts>Nonlinear>Arc-Length Opts

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ARCTRM, *Lab*, *VAL*, *NODE*, *DOF*

Controls termination of the solution when the arc-length method is used.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Specifies the basis of solution termination:

OFF

Does not use **ARCTRM (p. 177)** to terminate analysis (default).

L

Terminates the analysis if the first limit point has been reached. The first limit point is that point in the response history when the tangent stiffness matrix becomes singular (that is, the point at which the structure becomes unstable). If *Lab* = L, arguments *VAL*, *NODE*, *DOF* are ignored.

U

Terminates the analysis when the displacement first equals or exceeds the maximum desired value.

VAL

Maximum desired displacement (absolute value). Valid only if *Lab* = U. The analysis terminates whenever the calculated displacement first equals or exceeds this value. For rotational degrees of freedom, *VAL* must be in radians (not degrees).

NODE

Node number corresponding to displacement used to compare with displacement specified by *VAL*. If blank, the maximum displacement will be used. Valid only if *Lab* = U.

DOF

Valid degree of freedom label for nodal displacement specified by *NODE*. Valid labels are UX, UY, UZ, ROTX, ROTY, ROTZ. Valid only if *NODE*>0 and *Lab* = U.

Notes

The **ARCTRM (p. 177)** command is valid only when the arc-length method (**ARCLEN (p. 175)**,ON) is used.

It can be convenient to use this command to terminate the analysis when the first limit point is reached. In addition, the **NCNV (p. 1210)** command should be used to limit the maximum number of iterations. If the **ARCTRM (p. 177)** command is not used, and the applied load is so large that the solution path can never reach that load, the arc-length solution will continue to run until a CPU time limit or a "maximum number of iterations" is reached.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Arc-Length Opts

Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL

Main Menu>Solution>Load Step Opts>Nonlinear>Arc-Length Opts

AREAS

Specifies "Areas" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

AREFINE, *NA1*, *NA2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified areas.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

Areas (*NA1* to *NA2* in increments of *NINC*) around which the mesh is to be refined. *NA2* defaults to *NA1*, and *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and all selected areas are used for refinement. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEPTH

Depth of mesh refinement in terms of the number of elements outward from the indicated areas (defaults to 1).

POST

Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF

No postprocessing will be done.

SMOOTH

Smoothing will be done. Node locations may change.

CLEAN

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The Mechanical APDL program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

OFF

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

AREFINE (p. 178) performs local mesh refinement around the specified areas. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

AREFINE (p. 178) refines all area elements and tetrahedral volume elements that are adjacent to the specified areas. Any volume elements that are adjacent to the specified areas, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes (**IC (p. 921)**), coupled nodes (**CP (p. 421)** family of commands), constraint equations (**CE (p. 297)** family of commands), or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. See Revising Your Model in the *Modeling and Meshing Guide* for additional restrictions on mesh refinement.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Areas

AREMESH, *LCOMB*, *ANGLE*

Generates an area in which to create a new mesh for rezoning.

SOLUTION (p. 38): Rezoning (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCOMB

Specifies how to combine adjacent line segments:

0

Line segments combined by connecting ends to ends. This value is the default.

-1

No line segments combined.

ANGLE

The maximum angle (in degrees) allowed for connecting two line segments together. The default value is 30. This value is valid only when *LCOMB* = 0.

Notes

Issue the **AREMESH (p. 180)** command *after* issuing a **REMESH (p. 1581)**,START command and *before* issuing a **REMESH (p. 1581)**,FINISH command.

The **AREMESH (p. 180)** command cannot account for an open area (or "hole") inside a completely enclosed region. Instead, try meshing around an open area by selecting two adjoining regions; for more information, see Hints for Remeshing Multiple Regions .

Menu Paths

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Create Rezone Area

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AREVERSE, *ANUM*, *NOEFLIP*

Reverses the normal of an area, regardless of its connectivity or mesh status.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ANUM

Area number of the area whose normal is to be reversed. If *ANUM* = ALL, the normals of all selected areas will be reversed. If *ANUM* = P, graphical picking is enabled. A component name may also be substituted for *ANUM*.

NOEFLIP

Indicates whether you want to change the normal direction of the existing elements on the reversed area(s) so that they are consistent with each area's new normal direction.

0

Make the normal direction of existing elements on the reversed area(s) consistent with each area's new normal direction (default).

1

Do not change the normal direction of existing elements on the reversed area(s).

Notes

You cannot use the **AREVERSE (p. 181)** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable. Also, you cannot use this command to change the normal direction for areas attached to volumes because IGES and ANF data is unchanged by reversal. Reversed areas that are attached to volumes need to be reversed again when imported.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

See Revising Your Model in the *Modeling and Meshing Guide* for more information.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AROTAT, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *PAX1*, *PAX2*, *ARC*, *NSEG*

Generates cylindrical areas by rotating a line pattern about an axis.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NL3, . . . , NL6

List of lines in the pattern to be rotated (6 maximum if using keyboard entry of *NL1* to *NL6*). The lines must lie in the plane of the axis of rotation. If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). If *NL1* = ALL, all selected lines will define the pattern to be rotated. A component name may also be substituted for *NL1*.

PAX1, PAX2

Keypoints defining the axis about which the line pattern is to be rotated.

ARC

Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360°.

NSEG

Number of areas (8 maximum) around circumference. Defaults to minimum number required for 90° -maximum arcs, that is, 4 for 360°, 3 for 270°, etc.

Notes

Generates cylindrical areas (and their corresponding keypoints and lines) by rotating a line pattern (and its associated keypoint pattern) about an axis. Keypoint patterns are generated at regular angular locations, based on a maximum spacing of 90°. Line patterns are generated at the keypoint patterns. Arc lines are also generated to connect the keypoints circumferentially. Keypoint, line, and area numbers are automatically assigned, beginning with the lowest available values (**NUMSTR (p. 1301)**). Adjacent lines use a common keypoint. Adjacent areas use a common line.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Lines>About Axis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ARSCALE, *NA1*, *NA2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of areas from a pattern of areas.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

Set of areas, *NA1* to *NA2* in steps of *NINC*, that defines the pattern to be scaled. *NA2* defaults to *NA1*, *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is defined by all selected areas. If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

RX*, *RY*, *RZ

Scale factors to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system. (*RR*, *R* θ, *RZ* for cylindrical; *RR*, *R* θ, *R* Φ for spherical). Note that the *R* θ and *R* Φ scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Nodes and elements associated with the original areas will be generated (scaled) if they exist.

1

Nodes and elements will *not* be generated.

IMOVE

Specifies whether areas will be moved or newly defined:

0

Additional areas will be generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Original areas will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a scaled set of areas (and their corresponding keypoints, lines, and mesh) from a pattern of areas. The MAT, TYPE, REAL, and ESYS attributes are based on the areas in the pattern and not the current settings. Scaling is done in the active coordinate system. Areas in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Areas

ARSYM, *Ncomp*, *NA1*, *NA2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates areas from an area pattern by symmetry reflection.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ncomp

Symmetry key:

X

X symmetry (default).

Y

Y symmetry.

Z

Z symmetry.

NA1*, *NA2*, *NINC

Reflect areas from pattern beginning with *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas (**ASEL (p. 192)**). If *Ncomp* = P, use graphical picking to specify areas and ignore *NL2* and *NINC*. A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Generate nodes and elements associated with the original areas, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether areas will be moved or newly defined:

0

Generate additional areas.

1

Move original areas to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of areas (and their corresponding keypoints, lines and mesh) from a given area pattern by a symmetry reflection (see analogous node symmetry command, **NSYM (p. 1289)**). The MAT, TYPE, REAL, ESYS, and SECNUM attributes are based upon the areas in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be a Cartesian system. Areas in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Areas are generated as described in the **AGEN (p. 107)** command.

See the **ESYM (p. 684)** command for additional information about symmetry elements.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASBA, *NA1*, *NA2*, *SEP0*, *KEEP1*, *KEEP2*

Subtracts areas from areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1

Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. Areas specified in this argument are not available for use in the *NA2* argument. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA1*.

NA2

Area (or areas, if picking is used) to subtract. If ALL, use all selected areas (except those included in the *NA1* argument). A component name may also be substituted for *NA2*.

SEP0

Behavior if the intersection of the *NA1* areas and the *NA2* areas is a line or lines:

(blank)

The resulting areas will share line(s) where they touch.

SEPO

The resulting areas will have separate, but coincident line(s) where they touch.

KEEP1

Specifies whether *NA1* areas are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NA1* areas after **ASBA (p. 186)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NA1* areas after **ASBA (p. 186)** operation (override **BOPTN (p. 269)** command settings).

KEEP2

Specifies whether *NA2* areas are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELETE

Delete *NA2* areas after **ASBA (p. 186)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NA2* areas after **ASBA (p. 186)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new areas by subtracting the regions common to both *NA1* and *NA2* areas (the intersection) from the *NA1* areas. The intersection can be an area(s) or line(s). If the intersection is a line and *SEP0* is blank, the *NA1* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEP0* is set to SEPO, *NA1* is divided into two unconnected areas with separate lines where they touch. See Solid Modeling in the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **ASBA (p. 186)**,ALL,ALL will have no effect since all the areas (in *NA1*) will be unavailable as *NA2* areas.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Area

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Area

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Areas

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Areas

---

ASBL, *NA*, *NL*, - - , *KEEPA*, *KEEPL*

Subtracts lines from areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA

Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA*.

NL

Line (or lines, if picking is used) to subtract. If ALL, use all selected lines. A component name may also be substituted for *NL*.

- -

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEEPA

Specifies whether *NA* areas are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NA* areas after **ASBL (p. 187)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NA* areas after **ASBL (p. 187)** operation (override **BOPTN (p. 269)** command settings).

KEEPL

Specifies whether *NL* lines are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NL* lines after **ASBL (p. 187)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NL* lines after **ASBL (p. 187)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new areas by subtracting the regions common to both the areas and lines (the intersection) from the *NA* areas. The intersection will be a line(s). See Solid Modeling in the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Line

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Line

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASBV, *NA*, *NV*, *SEPO*, *KEEPA*, *KEEPV*

Subtracts volumes from areas.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA

Area (or areas, if picking is used) to be subtracted from. If ALL, use all selected areas. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NA*.

NV

Volume (or volumes, if picking is used) to subtract. If ALL, use all selected volumes. A component name may also be substituted for *NV*.

SEPO

Behavior if the intersection of the areas and the volumes is a line or lines:

(blank)

The resulting areas will share line(s) where they touch.

SEPO

The resulting areas will have separate, but coincident line(s) where they touch.

KEEPA

Specifies whether *NA* areas are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NA* areas after **ASBV (p. 189)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NA* areas after **ASBV (p. 189)** operation (override **BOPTN (p. 269)** command settings).

KEEPV

Specifies whether *NV* volumes are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELETE

Delete volumes after **ASBV (p. 189)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep volumes after **ASBV (p. 189)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new areas by subtracting the regions common to both *NA* areas and *NV* volumes (the intersection) from the *NA* areas. The intersection can be an area(s) or line(s). If the intersection is a line and *SEPO* is blank, the *NA* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEPO* is set to SEPO, *NA* is divided into two unconnected areas with separate lines where they touch. See Solid Modeling in the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by Volume

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by Volume

Main Menu>Preprocessor>Modeling>Operate>Divide>Area by Volu

ASBW, *NA*, *SEPO*, *KEEP*

Subtracts the intersection of the working plane from areas (divides areas).

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA

Area (or areas, if picking is used) to be subtracted from. If *NA* = ALL, use all selected areas. If *NA* = P, graphical picking is enabled (valid only in the GUI). A component name may also be input for *NA*.

SEPO

Behavior of the created boundary.

(blank)

The resulting areas will share line(s) where they touch.

SEPO

The resulting areas will have separate, but coincident line(s).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEEP

Specifies whether *NA* areas are to be deleted.

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NA* areas after **ASBW (p. 190)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NA* areas after **ASBW (p. 190)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new areas by subtracting the intersection of the working plane from the *NA* areas. The intersection will be a line(s). The working plane must not be in the same plane as the *NA* area(s). If *SEPO* is blank, the *NA* area is divided at the line and the resulting areas will be connected, sharing a common line where they touch. If *SEPO* is set to SEPO, *NA* is divided into two unconnected areas with separate lines. The SEPO option may cause unintended consequences if any keypoints exist along the cut plane. See Solid Modeling in the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Issuing the **ASBW (p. 190)** command under certain conditions may generate a topological degeneracy error. Do not issue the command if:

- A sphere or cylinder has been scaled. (A cylinder must be scaled unevenly in the XY plane.)
- A sphere or cylinder has not been scaled but the work plane has been rotated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Area by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Area by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Divide>Area by WrkPlane

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASCRES, *Opt*

Specifies the output type for an acoustic scattering analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Opt

Output option:

TOTAL

Output the total pressure field (default).

SCAT

Output the scattered pressure field.

Notes

Use the **ASCRES (p. 192)** command to specify the output type for an acoustic scattering analysis.

The scattered option (*Opt* = SCAT) provides a scattered pressure output, p_{sc}, required for calculating target strength (TS).

The default behavior (*Opt* = TOTAL) provides a sum of the incident and scattering fields, $p^{total} = p^{inc} + p^{sc}$.

Issue the **AWAVE (p. 213)** command to define the incident pressure p^{inc}. If the **AWAVE (p. 213)** command is defined with *Opt2* = INT, only the total pressure field is output regardless of the **ASCRES (p. 192)**,*Opt* command.

Menu Paths

This command cannot be accessed from a menu.

ASEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

Selects a subset of areas.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

S

Select a new set (default)

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item

Label identifying data. Valid item labels are shown in Table 103: ASEL - Valid Item and Component Labels (p. 195). Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to AREA.

Comp

Component of the item (if required). Valid component labels are shown in Table 103: ASEL - Valid Item and Component Labels (p. 195).

VMIN

Minimum value of item range. Ranges are area numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM (p. 356)** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **ALIST (p. 113)** command for a discussion of signed attributes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN*.

VINC

Value increment within range. Used only with integer ranges (such as for area numbers). Defaults to 1. *VINC* cannot be negative.

KSWP

Specifies whether only areas are to be selected:

0

Select areas only.

1

Select areas, as well as keypoints, lines, nodes, and elements associated with selected areas. Valid only with *Type* = S.

Command Default

All areas are selected.

Notes

Selects a subset of areas. For example, to select those areas with area numbers 1 through 7, use **ASEL (p. 192)**,S,AREA,,1,7. The selected subset is then used when the ALL label is entered (or implied) on other commands, such as **ALIST (p. 113)**,ALL. Only data identified by area number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

In a cyclic symmetry analysis, area hot spots can be modified. Consequently, the result of an area selection may be different before and after the **CYCLIC (p. 462)** command.

If *Item* = ACCA, the command selects only those areas that were created by concatenation. The *KSWP* field is processed, but the *Comp*, *VMIN*, *VMAX*, and *VINC* fields are ignored.

This command is valid in any processor.

For Selects based on non-integer numbers (coordinates, results, etc.), items that are within the range VMIN-*Toler* and VMAX+*Toler* are selected. The default tolerance *Toler* is based on the relative values of VMIN and VMAX as follows:

- If VMIN = VMAX, *Toler* = 0.005 x VMIN.
- If VMIN = VMAX = 0.0, *Toler* = 1.0E-6.
- If VMAX ≠ VMIN, *Toler* = 1.0E-8 x (VMAX-VMIN).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the **SELTOL (p. 1710)** (p. 1710) command to override this default and specify *Toler* explicitly.

Table 103: ASEL - Valid Item and Component Labels

Valid Item and Component Labels ASEL (p. 192), *Type*	*Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*	
Item	**Comp**	**Description**
AREA		Area number.
EXT		Area numbers on exterior of selected volumes (ignore remaining fields).
LOC	X, Y, Z	X, Y, or Z center (picking "hot spot" location in the active coordinate system).
HPT		Area number (selects only areas with associated hard points).
MAT		Material number associated with the area.
TYPE		Element type number associated with the area.
REAL		Real constant set number associated with the area.
ESYS		Element coordinate system associated with the area.
SECN		Section number associated with the area.
ACCA		Concatenated areas (selects only areas that were created by area concatenation (**ACCAT (p. 90)**)).

Menu Paths

Utility Menu>Select>Entities

ASIFILE, *Opt*, *Fname*, *Ext*, *Oper*, *kDim*, *kOut*, *LIMIT*, --, *ResOpt*

Writes or reads one-way acoustic-structural coupling data.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Opt

Command behavior option:

WRITE

Write the structural results to the specified file.

READ

Read the structural results from the specified file. This option is invalid during **/POST1 (p. 1461)** postprocessing.

Fname

File name and directory path of a one-way acoustic-structural coupling data file (248 characters maximum, including the characters needed for the directory path). An unspecified directory path

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defaults to the working directory; in this case, you can use all 248 characters for the file name (defaults to `jobname`).

Ext

File name extension of the one-way acoustic-structural coupling data file (defaults to `.asi`).

Oper

Command operation:

NOMAP

No mapping occurs between the structural and acoustic models when reading the structural results from the specified file (default).

MAP

Maps the results from the structural to the acoustic model. (See "Notes" (p. 197).)

kDim

Interpolation criteria. Valid only when *Oper* = MAP.

If *kDim* = 2 or 0, two-dimensional interpolation is applied (that is, interpolate occurs on a surface).

kOut

Outside region results. Valid only when *Oper* = MAP.

If *kOut* = 0, use the value(s) of the nearest region point for points outside of the region.

If *kOut* = 1, set results extrapolated outside of the region to zero.

LIMIT

Number of nearby nodes considered for interpolation. Valid only when *Oper* = MAP.

Minimum = 5. Default = 20.

Lower values reduce processing time; however, some distorted or irregular meshes require a higher value to encounter three nodes for triangulation.

--

Unused field.

ResOpt

Transient results option:

ACEL

Output or input the particle acceleration in a transient analysis (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VELO

Output or input the particle velocity in a transient analysis.

Notes

The **ASIFILE (p. 195)** command writes to, or reads from, a file containing one-way acoustic-structural coupling data.

Results data on the one-way coupling interface (defined by the **SF (p. 1733)**,,FSIN command) in the structural model are written to the one-way coupling result data file during the structural solution.

By default, one-way coupling results data are read into the acoustic model as the velocity (harmonic) or acceleration (transient) excitation during the sequential acoustic solution. If the transient is to be solved with the velocity potential formulation in acoustics, set *ResOpt* = VELO to write/read the results data as velocity excitation.

If *Oper* = NOMAP, both structural and acoustic models must share the same node number on the one-way coupling interface.

If *Oper* = MAP:

- The one-way coupling interface must be defined in the acoustic model (**SF (p. 1733)**,,FSIN) such that it corresponds to the field-surface interface number (FSIN) in the structural model.
- The output points are correct only if they are within the boundaries set via the specified input points.
- Calculations for out-of-bound points require much more processing time than do points that are within bounds.
- For each point in the acoustic destination mesh, the command searches all possible triangles in the structural source mesh to find the best triangle containing each point, then performs a linear interpolation inside this triangle. For faster and more accurate results, consider your interpolation method and search criteria carefully (see *LIMIT*).

You can also write an `.asi` file during postprocessing of the structural model. In the POST1 postprocessor (**/POST1 (p. 1461)**), issue the command **ASIFILE (p. 195)**,WRITE,*Fname*,*Ext* to output results on selected surface nodes of the structural model. In the subsequent acoustic analysis, apply the **SF (p. 1733)**,,FSIN,1 command on the selected nodes of the acoustic model, and issue the **ASIFILE (p. 195)**,READ,*Fname*,*Ext* command to read the `.asi` file.

One-way coupling excitation can be applied to multiple frequencies or time steps.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASKIN, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Generates an area by "skinning" a surface through guiding lines.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

The first guiding line forming the skinned area. If *NL1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NL1*. If *NL1* is negative, the line beginnings and ends will be used to direct the skinning of the remaining lines (see "Changing the ASKIN Algorithm" (p. 198) below).

NL2, NL3, NL4, . . . , NL9

The additional guiding lines for the skinned area (up to 9 total lines, including *NL1*, if using keyboard entry). If negative (and *NL1* is negative), the line beginning and end will be temporarily interchanged for the skinning operation (see "Changing the ASKIN Algorithm" (p. 198) below).

Notes

Generates an area by "skinning" a surface through specified guiding lines. The lines act as a set of "ribs" over which a surface is "stretched." Two opposite edges of the area are framed by the first (*NL1*) and last (*NLn*) guiding lines specified. The other two edges of the area are framed by splines-fit lines which the program automatically generates through the ends of all guiding lines. The interior of the area is shaped by the interior guiding lines. Once the area has been created, only the four edge lines will be attached to it. In rare cases, it may be necessary to change the default algorithm used by the **ASKIN (p. 198)** command (see "Changing the ASKIN Algorithm" (p. 198) below).

Changing the **ASKIN (p. 198)** Algorithm

When skinning from one guiding line to the next, the program can create the transition area in one of two ways: one more spiraled and one less spiraled ("flatter"). By default, the program attempts to produce the flatter transition, instead of the more spiraled transition. This algorithm can be changed by inputting *NL1* as a negative number, in which case the program connects all the keypoints at the line "beginnings" (**/PSYMB (p. 1552)**,LDIR command) as one edge of the area, and all the line "ends" as the opposite edge, irrespective of the amount of spiraling produced in each transition area.

To further control the geometry of the area (if *NL1* is negative), the beginning and end of any specified line (other than *NL1*) can be temporarily interchanged (for the skinning operation only) by inputting that line number as negative. See Solid Modeling in the *Modeling and Meshing Guide* for an illustration.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>By Skinning

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASLL, *Type*, *ARKEY*

Selects those areas containing the selected lines.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of area select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

ARKEY

Specifies whether all contained area lines must be selected (**LSEL (p. 1074)**):

0

Select area if any of its lines are in the selected line set.

1

Select area only if all of its lines are in the selected line set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASLV, *Type*

Selects those areas contained in the selected volumes.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of area select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ASOL, *Lab*, *Opt*

Specifies the acoustic solver with scattered field formulation.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Acoustic solver specification (no default):

SCAT

Set acoustic solver to the scattered field formulation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Opt

Option identifying an acoustic solver status:

OFF

Deactivate the specified acoustic solver (default).

ON

Activate the specified acoustic solver.

Notes

Use the **ASOL (p. 200)** command to activate the specified acoustic solution process.

The scattered option (*Lab* = SCAT) sets the acoustic solver to the scattered-pressure field formulation.

Issue the **AWAVE (p. 213)** command to define the incident pressure p^{inc}. If the **AWAVE (p. 213)** command is defined with *Opt2* = INT, the acoustic solver is set to the scattered field formulation regardless of the **ASOL (p. 200)** command issued.

Menu Paths

This command cannot be accessed from a menu.

/**ASSIGN**,*Ident*,*Fname*,*Ext*,*--*,*LGkey*

Reassigns a file name to a Mechanical APDL file identifier.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ident

Mechanical APDL file name identifier. Valid identifiers are: cms, emat, erot, esav, full, LN07, LN09, LN11, LN20, LN21, LN22, LN25, LN31, LN32, mode, osav, rdsp , rmg, rst, rstp, rth, seld, and sscr. See File Management and Files for file descriptions. If blank, list currently reassigned files.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LGkey

Key to specify local or global file name control for the specified file identifier in a distributed-memory parallel processing run. For more information on local and global files, see File Handling Conventions in the *Parallel Processing Guide*.

BOTH

Reassign the file name for both the local and global files (default).

LOCAL

Reassign the file name for only the local files.

GLOBAL

Reassign the file name for only the global file.

Notes

The reassignment of file names is valid only if it is done before the file is used. All file reassignments are retained (not cleared) even if the database is cleared (**/CLEAR (p. 350)**) or the Jobname is changed (**/FILNAME (p. 757)**). Assigned files may be overwritten. If file name arguments (*Fname*, *Ext*, `--`) are blank, the default Mechanical APDL assignment is restored. Use **SEOPT (p. 1715)** for SUB files and **SEEXP (p. 1705)** for DSUB files.

This command is valid only at the Begin level.

This command also checks to ensure that the path/file is valid and can be written by the user. If it is not valid, an error message will be returned. Ensure that the directory exists prior to using **/ASSIGN (p. 201)** command.

Menu Paths

Utility Menu>File>File Options

ASUB, *NA1*, *P1*, *P2*, *P3*, *P4*

Generates an area using the shape of an existing area.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1

Existing area number whose shape is to be used. If *P1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI).

P1

Keypoint defining starting corner of area.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P2

Keypoint defining second corner of area.

P3

Keypoint defining third corner of area.

P4

Keypoint defining fourth corner of area (defaults to *P3*).

Notes

The new area will overlay the old area. Often used when the area to be subdivided consists of a complex shape that was not generated in a single coordinate system. Keypoints and any corresponding lines must lie on the existing area. Missing lines are generated to lie on the given area. The active coordinate system is ignored.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Arbitrary>Overlaid on Area

---

ASUM, *LAB*

Calculates and prints geometry statistics of the selected areas.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LAB

Controls the degree of tessellation used in the calculation of area properties. If *LAB* = DEFAULT, area calculations will use the degree of tessellation set through the **/FACET (p. 731)** command. If *LAB* = FINE, area calculations are based on a finer tessellation.

Notes

Calculates and prints geometry statistics (area, centroid location, moments of inertia, volume, etc.) associated with the selected areas. **ASUM (p. 203)** should only be used on perfectly flat areas.

Geometry items are reported in the global Cartesian coordinate system. A unit thickness is assumed unless the areas have a non-zero total thickness defined by real constant or section data.

For layered areas, a unit density is always assumed. For single-layer areas, a unit density is assumed unless the areas have a valid material (density).

The thickness and density are associated to the areas via the **AATT (p. 85)** command.

Items calculated via **ASUM (p. 203)** and later retrieved via a ***GET (p. 797)** or ***VGET (p. 2118)** command are valid only if the model is not modified after issuing the **ASUM (p. 203)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting a finer degree of tessellation will provide area calculations with greater accuracy, especially for thin, hollow models. However, using a finer degree of tessellation requires longer processing.

For very narrow (sliver) areas, such that the ratio of the minimum to the maximum dimension is less than 0.01, the **ASUM (p. 203)** command can provide erroneous area information. To ensure that the calculations are accurate, subdivide such areas so that the ratio of the minimum to the maximum is at least 0.05.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Areas

---

ATAN, *IR*, *IA*, - - , - - , *Name*, - - , - - , *FACTA*

Forms the arctangent of a complex variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the complex variable to be operated on.

- -, - -

Unused fields.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

- -, - -

Unused fields.

FACTA

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0). Usually *FACTA* should be set to 1. *FACTA* may affect the position of the angle by a multiple of π, resulting in a quadrant change.

Notes

Forms the arctangent of a complex variable according to the operation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IR = ATAN(*FACTA* X *b*/*a*)

where *a* and *b* are the real and imaginary parts, respectively, of the complex variable *IA* (which is of the form *a* + *ib*). The arctangent represents the phase angle (in radians), and is valid only for a harmonic analysis (**ANTYPE (p. 162)**,HARMIC).

Since the scaling factor is applied uniformly to *b*/*a*, applying *any* positive or negative scaling factor will not affect the size of the phase angle, with the exception that a negative scaling factor will change the results quadrant by π. The magnitude of a complex number is still obtained through the **ABS (p. 89)** command. See POST26 - Data Operations in the *Mechanical APDL Theory Reference* for details.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Arctangent

ATRAN, *KCNTO*, *NA1*, *NA2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of areas to another coordinate system.

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCNTO

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

NA1, NA2, NINC

Transfer area pattern beginning with *NA1* to *NA2* (defaults to *NA1*) in steps of *NINC* (defaults to 1). If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is all selected areas (**ASEL (p. 192)**). If *NA1* = P, graphical picking is enabled and all remaining arguments are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original areas, if they exist.

1

Do not generate nodes and elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IMOVE

Specifies whether to redefine the existing areas:

0

Generate additional areas.

1

Move original areas to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old areas are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of areas (and their corresponding lines, keypoints and mesh) from one coordinate system to another (see analogous node **TRANSFER (p. 2011)** command). The MAT, TYPE, REAL, and ESYS attributes are based upon the areas in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Areas are generated as described in the **AGEN (p. 107)** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Areas

---

ATYPE

Specifies "Analysis types" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Analysis Type

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/AUTO, *WN*

Resets the focus and distance specifications to "automatically calculated."

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

Notes

Focus point and distance will be automatically calculated during next display. Settings may still be changed with the **/FOCUS (p. 771)** and **/DIST (p. 536)** commands after this command has been issued. See also the **/USER (p. 2061)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Automatic Fit Mode

AUTOTS, *Key*

Specifies whether to use automatic time stepping or load stepping.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Automatic time stepping key:

OFF

Do not use automatic time stepping.

ON

Use automatic time stepping (default).

AUTO

The program determines whether to use automatic time stepping (used by Workbench).

Command Default

Automatic time stepping is on by default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Specifies whether to use automatic time stepping (or load stepping) over this load step. If *Key* = ON, both time step prediction and time step bisection will be used.

Bisection does not occur with **THOPT (p. 1997)**,QUASI since it uses only one equilibrium iteration per substep. To ensure bisection, use the iterative QUASI method (**THOPT (p. 1997)**,QUASI,,,,,,1).

You cannot use automatic time stepping (**AUTOTS (p. 207)**), line search (**LNSRCH (p. 1057)**), or the DOF solution predictor (**PRED (p. 1476)**) with the arc-length method (**ARCLEN (p. 175)**, **ARCTRM (p. 177)**). If you activate the arc-length method after you set **AUTOTS (p. 207)**, **LNSRCH (p. 1057)**, or **PRED (p. 1476)**, a warning message appears. If you choose to proceed with the arc-length method, the program disables your automatic time stepping, line search, and DOF predictor settings, and the time step size is controlled by the arc-length method internally.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substeps

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substeps

---

/AUX2

Enters the binary file dumping processor.

AUX2 (p. 61): Binary File Dump (p. 61)
SESSION (p. 11): Processor Entry (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the binary file-dumping processor (auxiliary processor AUX2), used for dumping the contents of certain Mechanical APDL binary files for visual examination.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>List>Binary Files

Utility Menu>List>Files>Binary Files

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/AUX3

Enters the results file editing processor.

AUX3 (p. 61): Results Files (p. 61)
SESSION (p. 11): Processor Entry (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the results-file editing processor (auxiliary processor AUX3), used for editing Mechanical APDL results files.

A pending **DELETE (p. 513)** is processed when **FINISH (p. 758)** or **/EOF (p. 645)** is issued. To cancel a pending **DELETE (p. 513)**, issue **UNDELETE (p. 2052)**.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

/AUX12

Enters the radiation processor.

AUX12 (p. 62): General Radiation (p. 62)
SESSION (p. 11): Processor Entry (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the radiation processor (auxiliary processor AUX12). This processor supports the Radiation Matrix and the Radiosity Solver methods.

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Radiation Matrix

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/AUX15

Enters the IGES file transfer processor.

AUX15 (p. 63): IGES (p. 63)
POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the IGES file-transfer processor (auxiliary processor AUX15), used to read an IGES data file into the Mechanical APDL program.

This command is valid only at the Begin Level.

Menu Paths

Utility Menu>File>Import

AVPRIN, *KEY*, *EFFNU*

Specifies how principal and vector sums are to be calculated.

POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Averaging key:

0

Average the component values from the elements at a common node, then calculate the principal or vector sum from the averaged components (default).

1

Calculate the principal or vector sum values on a per element basis, then average these values from the elements at a common node.

EFFNU

Effective Poisson's ratio used for computing the von Mises equivalent strain (EQV). This command option is intended for use with line elements or in load case operations (**LCOPER (p. 1026)**); the program automatically selects the most appropriate effective Poisson's ratio, as discussed below.

Command Default

Average components at common node before principal or vector sum calculation except for the von Mises equivalent strain (EQV), see below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Selects the method of combining components for certain derived nodal results when two or more elements connect to a common node. The methods apply to the calculations of derived nodal principal stresses, principal strains, and vector sums for selects, sorts, and output (**NSEL (p. 1266)**, **NSORT (p. 1283)**, **PRNSOL (p. 1505)**, **PLNSOL (p. 1425)**, etc.). The calculation of these nodal results excludes beam and pipe elements.

This command also defines the effective Poisson's ratio (*EFFNU*) used for equivalent strain calculations. If you use *EFFNU*, the default effective Poisson's ratios shown below will be overridden for all elements by the *EFFNU* value. To return to the default settings, issue the **RESET (p. 1598)** command. The default value for *EFFNU* is:

- Poisson's ratio as defined on the **MP (p. 1160)** commands for EPEL and EPTH
- 0.5 for EPPL and EPCR
- 0.5 if the referenced material is hyperelastic
- 0.0 for line elements (includes beam, link, and pipe elements, as well as discrete elements), cyclic symmetry analysis, mode superposition analyses (with *MSUPkey* = YES on the **MXPAND (p. 1203)** command), and load case operations (**LCOPER (p. 1026)**).
- For multistage analysis, a value of *EFFNU* must always be specified.

If *EFFNU* is specified, the calculation of von Mises equivalent strain (EQV) is performed according to the *KEY* setting. However, if *EFFNU* is not specified, or if the **AVPRIN (p. 210)** command is not issued, the von Mises equivalent strain is calculated using the average of the equivalent strains from the elements at a common node (behavior of *KEY* =1) irrespective of the value of the averaging KEY.

For a random vibration (PSD) analysis (for more details, see Review the Results in the *Structural Analysis Guide*):

- Issuing either **AVPRIN (p. 210)**,0 or **AVPRIN (p. 210)**,1 calculates the principal stresses using the appropriate averaging method. They are then used to determine SEQV. The output will have non-zero values for the principal stresses.
- If **AVPRIN (p. 210)** is not issued, the Segalman-Fulcher method is used to calculate SEQV. This method does not calculate principal stresses, but directly calculates SEQV from the component stresses; therefore, the output will have zero values for the principal stresses.

This command is also valid in POST26, where applicable.

See Combined Stresses and Strains in the *Mechanical APDL Theory Reference* for more information.

Menu Paths

Main Menu>General Postproc>Element Table>Define Table

Main Menu>General Postproc>List Results>Nodal Solution

Main Menu>General Postproc>Options for Outp

Main Menu>General Postproc>Path Operations>Map onto Path

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>General Postproc>Path Operations>Map onto Path>FE Results

Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu

Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu

Utility Menu>List>Results>Nodal Solution

Utility Menu>List>Results>Options

Utility Menu>Plot>Results>Contour Plot>Elem Solution

Utility Menu>Plot>Results>Contour Plot>Nodal Solution

Utility Menu>PlotCtrls>Multi-Plot Contrls

AVRES, *KEY*, *Opt*

Specifies how results data will be averaged when PowerGraphics is enabled.

POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Averaging key.

1

Average results at all common subgrid locations.

2

Average results at all common subgrid locations except where material type (**MAT (p. 1111)**) discontinuities exist. (Default.)

3

Average results at all common subgrid locations except where real constant (**REAL (p. 1579)**) discontinuities exist.

4

Average results at all common subgrid locations except where material type (**MAT (p. 1111)**) or real constant (**REAL (p. 1579)**) discontinuities exist.

Opt

Option to determine how results data are averaged.

(blank)

Average surface results data using only the exterior element faces (default).

FULL

Average surface results data using the exterior face and interior element data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **AVRES (p. 212)** command specifies how results data will be averaged at subgrid locations that are common to 2 or more elements. The command is valid only when PowerGraphics is enabled (via the **/GRAPHICS (p. 866)**,POWER command).

With PowerGraphics active (**/GRAPHICS (p. 866)**,POWER), the averaging scheme for surface data with interior element data included (**AVRES (p. 212)**,,FULL) and multiple facets per edge (**/EFACET (p. 602)**,2 or **/EFACET (p. 602)**,4) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE (p. 2034)**,,6 or **/TYPE (p. 2034)**,,7). When the Section data is not included in the averaging schemes (**/TYPE (p. 2034)**,,7), the resulting absolute value for the midside node is significantly smaller.

PowerGraphics does not average your stresses across discontinuous surfaces. The normals for various planes and facets are compared to a tolerance to determine continuity. The *ANGLE* value you specify in the **/EDGE (p. 594)** command is the tolerance for classifying surfaces as continuous or "coplanar."

The command affects nodal solution contour plots (**PLNSOL (p. 1425)**), nodal solution printout (**PRNSOL (p. 1505)**), and subgrid solution results accessed through the Query Results function (under General Postprocessing) in the GUI.

The command has no effect on the nodal degree of freedom solution values (UX, UY, UZ, TEMP, etc.).

For cyclic symmetry mode-superposition harmonic solutions, **AVRES (p. 212)**,,FULL is not supported. Additionally, averaging does not occur across discontinuous surfaces, and the *ANGLE* value on the **/EDGE (p. 594)** command has no effect.

The section-based (**MAT (p. 1111)**) discontinuity in shells is accommodated via **/ESHAPE (p. 665)**.

The command is also available in **/SOLU (p. 1821)**.

Menu Paths

Main Menu>General Postproc>Options for Outp

Utility Menu>List>Results>Options

AWAVE,*Wavenum*,*Wavetype*,*Opt1*,*Opt2*,*VAL1*,*VAL2*,*VAL3*,*VAL4*,*VAL5*,*VAL6*,*VAL7*,*VAL8*,*VAL9*,*VAL10*,*VAL11*,*VAL12*,*VAL13*

Specifies input data for an acoustic incident wave.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Wavenum

Wave number. You specify the integer number for an acoustic incident wave inside or outside the model. The number must be between 1 and 20.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wavetype

Wave type:

PLAN

Planar incident wave

MONO

Monopole or pulsating sphere incident wave

DIPO

Dipole incident wave

BACK

Back enclosed loudspeaker

BARE

Bare loudspeaker

STATUS

Displays the status of the acoustic wave settings if *Wavenum* = a number between 1 and 20 or ALL.

DELE

Deletes the acoustic wave settings if *Wavenum* = a number between 1 and 20 or ALL.

Opt1

PRES

Pressure

VELO

Velocity

Opt2

EXT

Incident wave outside the model.

INT

Incident wave inside the model. This option is only available for pure scattered pressure formulation.

For *Wavetype* = PLAN, only *Opt2* = EXT is available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL1, VAL2, VAL3, . . . , VAL13

If *Wavetype* = PLAN, MONO, DIPO, BACK, or BARE:

VAL1

Amplitude of pressure or normal velocity to the sphere surface.

VAL2

Phase angle of the applied pressure or velocity (in degrees). Defaults to 0 degrees.

If *Wavetype* = PLAN:

VAL3

Incident φ angle from x axis toward y axis.

VAL4

Incident θ angle from z axis toward y axis.

VAL5

Not used.

If *Wavetype* = MONO, DIPO, BACK, or BARE:

VAL3 - VAL5

Global Cartesian coordinate values of source position.

If *Wavetype* = PLAN, MONO, DIPO, BACK, or BARE:

VAL6

Mass density of base medium (defaults to 1.2041 kg/m3).

VAL7

Speed of sound in base medium (defaults to 343.24 m/s).

VAL8

Radius of pulsating sphere (not used for *Wavetype* = PLAN).

VAL9

Dipole length (only available for *Wavetype* = DIPO, BARE).

VAL10 - VAL12

Unit vector of dipole axis from the positive to the negative. Only available for *Wavetype* = DIPO, BARE.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL13

Port number if the incident power is required on the port

Notes

Use the **ASOL (p. 200)** command to activate the scattered field algorithm and the **ASCRES (p. 192)** command for output control with the scattered field algorithm. Refer to Acoustics in the *Theory Reference* for more information about pure scattered field formulation.

Menu Paths

This command cannot be accessed from a menu.

/AXLAB,*Axis*,*Lab*

Labels the X and Y axes on graph displays.

GRAPHICS (p. 16): Graphs (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Axis

Axis specifier:

X

Apply label to X axis.

Y

Apply label to Y axis.

Lab

Axis label (user defined text up to 30 characters long). Leave blank to reestablish the default for *Axis* axis.

Command Default

Labels are determined by the program.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B Commands

/BATCH, *Lab*

Sets the program mode to "batch."

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Specifies listing mode during a batch run:

LIST

The batch output will include a listing of the input file.

(blank)

Suppress input data listing.

Command Default

Interactive mode.

Notes

Sets the program mode to "batch" when included as the first line on an input file of Mechanical APDL commands. For convenience, this command is automatically recorded on the log file (`Jobname.log`) at the beginning of an interactive Mechanical APDL session so that the log file can be reused later for batch input.

For instructions on starting a batch job from the command line in Linux and Windows, see Batch Mode in the *Operations Guide*.

To export graphics files in batch mode, you must issue the **/SHOW (p. 1785)** command before commands that produce a graphical output (for example, **PLNSOL (p. 1425)**, **PLESOL (p. 1401)**). For more details, see External Graphics Options in the *Basic Analysis Guide*.

Caution:

This command should not be entered directly in an interactive Ansys session since all subsequent keyboard input is simply copied to a file, without further processing or prompts from the program (use the "system break" to exit the Ansys program if this occurs).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The recommended method for choosing batch mode, rather than using the **/BATCH (p. 217)** command, is to select the **Batch** simulation environment from the **Product Launcher** task in the Ansys launcher, or the batch mode entry option on the Ansys execution command when entering the program.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

BCSOPTION, --,*Memory_Option*,*Memory_Size*,--,--,*Solve_Info*

Sets memory option for the sparse solver.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

--

Unused field

Memory_Option

Memory allocation option:

DEFAULT

Use the default memory allocation strategy for the sparse solver. The default strategy attempts to run in the INCORE memory mode. If there is not enough available physical memory when the solver starts to run in the INCORE memory mode, the solver will then attempt to run in the OUTOFCORE memory mode.

INCORE

Use a memory allocation strategy in the sparse solver that will attempt to obtain enough memory to run with the entire factorized matrix in memory. This option uses the most amount of memory and should avoid doing any I/O. By avoiding I/O, this option achieves optimal solver performance. However, a significant amount of memory is required to run in this mode, and it is only recommended on machines with a large amount of memory. If the allocation for in-core memory fails, the solver will automatically revert to out-of-core memory mode.

OUTOFCORE

Use a memory allocation strategy in the sparse solver that will attempt to allocate only enough work space to factor each individual frontal matrix in memory, but will store the entire factorized matrix on disk. Typically, this memory mode results in poor performance due to the potential bottleneck caused by the I/O to the various files written by the solver.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FORCE

This option, when used in conjunction with the *Memory_Size* option, allows you to force the sparse solver to run with a specific amount of memory. This option is only recommended for the advanced user who understands sparse solver memory requirements for the problem being solved, understands the physical memory on the system, and wants to control the sparse solver memory usage.

Memory_Size

Initial memory size allocation for the sparse solver in GB. This argument allows you to tune the sparse solver memory and is not generally required. Although there is no upper limit for *Memory_Size*, the *Memory_Size* setting should always be well within the physical memory available, but not so small as to cause the sparse solver to run out of memory. Warnings and/or errors from the sparse solver will appear if this value is set too low. If the FORCE memory option is used, this value is the amount of memory allocated for the entire duration of the sparse solver solution.

--,--

Unused fields

Solve_Info

Solver output option:

OFF

Turns off additional output printing from the sparse solver (default).

PERFORMANCE

Turns on additional output printing from the sparse solver, including a performance summary and a summary of file I/O for the sparse solver. Information on memory usage during assembly of the global matrix (that is, creation of the `Jobname.full` file) is also printed with this option.

Command Default

Automatic memory allocation is used.

Notes

This command controls options related to the sparse solver in all analysis types where the sparse solver can be used. It also controls the Block Lanczos eigensolver in a modal or buckling analysis.

The sparse solver runs from one large work space (that is, one large memory allocation). The amount of memory required for the sparse solver is unknown until the matrix structure is preprocessed, including equation reordering. The amount of memory allocated for the sparse solver is then dynamically adjusted to supply the solver what it needs to compute the solution.

If you have a very large memory system, you may want to try selecting the INCORE memory mode for larger jobs to improve performance. When running the sparse solver on a machine with very slow I/O

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

performance (for example, slow hard drive speed), you may want to try using the INCORE memory mode to achieve better performance. However, doing so may require much more memory compared to running in the OUTOFCORE memory mode.

Running with the INCORE memory mode is best for jobs which comfortably fit within the limits of the physical memory on a given system. If the sparse solver work space exceeds physical memory size, the system will be forced to use virtual memory (or the system page/swap file). In this case, it is typically more efficient to run with the OUTOFCORE memory mode. Assuming the job fits comfortably within the limits of the machine, running with the INCORE memory mode is often ideal for jobs where repeated solves are performed for a single matrix factorization. This occurs in a modal or buckling analysis or when doing multiple load steps in a linear, static analysis.

For repeated runs with the sparse solver, you may set the initial sparse solver memory allocation to the amount required for factorization. This strategy reduces the frequency of allocation and reallocation in the run to make the INCORE option fully effective. If you have a very large memory system, you may use the *Memory_Size* argument to increase the maximum size attempted for in-core runs.

Menu Paths

This command cannot be accessed from a menu.

BETAD, *VALUE*

Defines the stiffness matrix multiplier for damping.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VALUE

Stiffness matrix multiplier for damping.

Notes

This command defines the stiffness matrix multiplier β used to form the viscous damping matrix $[C]=\beta[K]$, where $[K]$ is the stiffness matrix.

Values of β can also be input as a material property (**MP (p. 1160)**,BETD or **TB (p. 1899)**,SDAMP,,,,BETD). If BETD in either form is included, the BETD value is added to the **BETAD (p. 220)** value as appropriate. (See Material Damping in the *Material Reference* and Damping Matrices in the *Theory Reference*.) Damping is not used in static (**ANTYPE (p. 162)**,STATIC) or buckling (**ANTYPE (p. 162)**,BUCKLE) analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping

Main Menu>Solution>Analysis Type>Sol'n Controls>Transient

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

BF, *Node*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *MESHFLAG*

Defines a nodal body-force load.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Node

Node to which body load applies. If *Node* = ALL, apply to all selected nodes (**NSEL (p. 1266)**). A component name may also be substituted for *Node*.

Lab

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	TEMP	Temperature
	FREQ	Frequency (harmonic analyses only)
	FLUE	Fluence
	FPBC	Floquet periodic boundary condition
Thermal	HGEN	Heat generation rate (updated by volume changes when large-deflection effects are included (**NLGEOM (p. 1231)**,ON))
	VELO	Velocity for mass transport
Magnetic	TEMP	Temperature
	VELO	Velocity or angular velocity
	MVDI	Magnetic virtual displacements flags
Electric	TEMP	Temperature
	CHRGD	Charge density
Acoustic	MASS	Mass source; mass source rate; or power source in an energy diffusion solution for room acoustics
	IMPD	Impedance sheet
	SPRE	Static pressure
	TEMP	Temperature
	VELO	Velocity or acceleration
	PORT	Interior acoustics port
	FPBC	Floquet periodic boundary condition
	VMEN	Mean flow velocity

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Body Load Label	Label Description
	UFOR	Force potential
	SFOR	Shear force for viscous-thermal acoustics or poroelastic acoustics
	HFLW	Volumetric heat source for viscous-thermal acoustics
Poromechanics	FSOU	Fluid flow source
Diffusion	TEMP	Temperature
	DGEN	Diffusing substance generation rate
	VELO	Transport velocity

VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6

Value associated with the *Lab* item or table name reference for tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, DGEN, MVDI, CHRGD, PORT, and SPRE.

Tabular input is supported for certain labels (see Notes (p. 228) for details). To specify a table, enclose the table name in percent signs (%) (e.g., **BF (p. 221)**,*Node*,TEMP,%*tabname*%). Use the ***DIM (p. 530)** command to define a table.

If *Lab* = MASS for acoustics:

VAL1

Mass source with units of kg/(m^3*s) in a harmonic analysis or in a transient analysis solved with the velocity potential formulation; or mass source rate with units of kg/(m^3*s^2) in a transient analysis solved with the pressure formulation; or power source with units of watts in an energy diffusion solution for room acoustics

VAL2

Phase angle in degrees

VAL3

Not used

VAL4

Not used

VAL5

Not used

VAL6

Not used

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *Lab* = VELO for acoustics (velocity components in a harmonic analysis or in a transient analysis solved with the velocity potential formulation; or acceleration components in a transient analysis solved with the pressure formulation):

VAL1

X component

VAL2

Y component

VAL3

Z component

VAL4

X-component phase angle in degrees

VAL5

Y-component phase angle in degrees

VAL6

Z-component phase angle in degrees

If *Lab* = VELO for electromagnetics (velocity and angular velocity components in the global Cartesian coordinate system):

VAL1

Velocity component in the X direction

VAL2

Velocity component in the Y direction

VAL3

Velocity component in the Z direction

VAL4

Angular velocity about the X axis (rad/sec)

VAL5

Angular velocity about the Y axis (rad/sec)

VAL6

Angular velocity about the Z axis (rad/sec)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *Lab* = VELO for thermal (velocity components in the global Cartesian coordinate system):

VAL1

Mass transport velocity component in X direction

VAL2

Mass transport velocity component in Y direction

VAL3

Mass transport velocity component in Z direction

VAL4

Not used

VAL5

Not used

VAL6

Not used

If *Lab* = VELO for diffusion (transport velocity):

VAL1

Transport velocity component in X direction

VAL2

Transport velocity component in Y direction

VAL3

Transport velocity component in Z direction

VAL4

Not used

VAL5

Not used

VAL6

Not used

If *Lab* = IMPD:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL1

Resistance in $N \cdot s/m^3$

VAL2

Reactance in $N \cdot s/m^3$

VAL3

Not used

VAL4

Not used

VAL5

Not used

VAL6

Not used

If *Lab* = FPBC:

VAL1

Phase shift (product of phase constant and period in unit radian); or Floquet boundary flag (set *VAL1* = YES) for a modal analysis that solves the eigenvalues with a specified frequency (*FREQMOD* on the **MODOPT (p. 1140)** command)

VAL2

Attenuation (product of attenuation constant and period); not used if *VAL1* = YES

VAL3

Not used

VAL4

Not used

VAL5

Not used

VAL6

Not used

If *Lab* = VMEN:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL1

Mean flow velocity component in the X direction

VAL2

Mean flow velocity component in the Y direction

VAL3

Mean flow velocity component in the Z direction

VAL4

Not used

VAL5

Not used

VAL6

Not used

If *Lab* = UFOR:

VAL1

Real part of complex force potential

VAL2

Imaginary part of complex force potential

VAL3

Not used

VAL4

Not used

VAL5

Not used

VAL6

Not used

If *Lab* = SFOR:

VAL1

X component of shear force for viscous-thermal acoustics or poroelastic acoustics

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL2

Y component of shear force for viscous-thermal acoustics or poroelastic acoustics

VAL3

Z component of shear force for viscous-thermal acoustics or poroelastic acoustics

VAL4

X-component phase angle in degrees

VAL5

Y-component phase angle in degrees

VAL6

Z-component phase angle in degrees

If *Lab* = HFLW:

VAL1

Real part of volumetric heat source for viscous-thermal acoustics

VAL2

Imaginary part of volumetric heat source for viscous-thermal acoustics

VAL3

Not used

VAL4

Not used

VAL5

Not used

VAL6

Not used

MESHFLAG

Specifies how to apply nodal body-force loading on the mesh. Valid in a nonlinear adaptivity analysis when *Lab* = HGEN or TEMP, and *Node* is not a component name.

0 – Nodal body-force loading occurs on the current mesh (default).

1 – Nodal body-force loading occurs on the initial mesh for nonlinear adaptivity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines a nodal body-force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.). Nodal body loads default to the **BFUNIF (p. 259)** values, if they were previously specified.

Table names are valid for *Lab* value (*VALn*) inputs in these cases only:

- *VAL1* = %*tabname*% for:

  temperatures (TEMP), diffusing substance generation rates (DGEN), and heat generation rates (HGEN).

- *VAL1* = %*tabname1*% and *VAL2* = %*tabname2*% for:

  mass source, mass source rate, or power source (MASS); the Floquet periodic boundary condition (FPBC); the force potential (UFOR); and the volumetric heat source (HFLW).

- *VAL1* = %*tabname1*%, *VAL2* = %*tabname2*%, and *VAL3* = %*tabname3*% for:

  mean flow velocities (VMEN).

- *VAL1* = %*tabname1*%, *VAL2* = %*tabname2*%, *VAL3* = %*tabname3*%, *VAL4* = %*tabname4*%, *VAL5* = %*tabname5*%, and *VAL6* = %*tabname6*% for:

  velocities or accelerations (VELO); and shear force (SFOR).

The heat generation rate loads specified with the **BF (p. 221)** command are multiplied by the weighted nodal volume of each element adjacent to that node. This yields the total heat generation at that node.

In a modal analysis, the Floquet periodic boundary condition (FPBC) is only valid for the acoustic elements FLUID30, FLUID220, and FLUID221.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

Graphical picking is available only via the listed menu paths.

Body load labels VELO, MASS, SPRE, FPBC, VMEN, UFOR, SFOR, and HFLW cannot be accessed from a menu.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Fluence>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Nodes

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Fluence>On Nodes

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Nodes

---

BFA,*Area*,*Lab*,*VAL1*,*VAL2*,*VAL3*,*VAL4*

Defines a body-force load on an area.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Area

Area to which body load applies. If ALL, apply to all selected areas (**ASEL (p. 192)**). A component name may also be substituted for *Area*.

Lab

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Body Load Label	Label Description
Magnetic	TEMP	temperature
	JS	current densities
Electric	TEMP	temperature
	CHRGD	charge density
Acoustic	IMPD	impedance sheet
	TEMP	temperature

VAL1, VAL2, VAL3

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, CHRGD. For *Lab* = JS in magnetics, use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components. For *Lab* = IMPD in acoustics, *VAL1* is the resistance and *VAL2* is the reactance in ohms/square. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFA (p. 229)**,*Area*,*Lab*,%*tabname*%. Use the ***DIM (p. 530)** command to define a table.

VAL4

If *Lab* = JS, *VAL4* is the phase angle in degrees.

Notes

Defines a body-force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on an area. Body loads may be transferred from areas to area elements (or to nodes if area elements do not exist) with the **BFTRAN (p. 259)** or **SBCTRAN (p. 1647)** commands. Body loads default to the value specified on the **BFUNIF (p. 259)** command, if it was previously specified.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFA (p. 229)** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Fluence>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Areas

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppMagField>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Fluence>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Areas

---

BFADELE,*AREA*,*Lab*

Deletes body-force loads on an area.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Area at which body load is to be deleted. If ALL, delete for all selected areas (**ASEL (p. 192)**). A component name may also be substituted for *AREA*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFA (p. 229)** command for labels.

Notes

Deletes body-force loads (and all corresponding finite element loads) for a specified area and label. Body loads may be defined on an area with the **BFA (p. 229)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Areas

Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Areas

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Areas

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Areas

Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Areas

Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Areas

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFALIST, *AREA*, *Lab*

Lists the body-force loads on an area.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Area at which body load is to be listed. If ALL (or blank), list for all selected areas (**ASEL (p. 192)**). If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFA (p. 229)** command for labels.

Notes

Lists the body-force loads for the specified area and label. Body loads may be defined on an area with the **BFA (p. 229)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Areas

Utility Menu>List>Loads>Body Loads>On Picked Areas

BFCUM, *Lab*, *Oper*, *FACT*, *TBASE*

Specifies that nodal body-force loads are to be accumulated.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Body Load Label	Label Description
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density
Diffusion	DGEN	diffusing substance generation rate

Oper

Accumulation key:

REPL

Subsequent values replace the previous values (default).

ADD

Subsequent values are added to the previous values.

IGNO

Subsequent values are ignored.

FACT

Scale factor for the nodal body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

TBASE

Used (only with *Lab* = TEMP) to calculate the temperature used in the add or replace operation (see *Oper*) as:

Temperature = *TBASE* + *FACT** (*T* - *TBASE*)

where *T* is the temperature specified on subsequent **BF (p. 221)** commands. *TBASE* defaults to zero.

Command Default

Replace previous values.

Notes

Allows repeated nodal body-force loads to be replaced, added, or ignored. Nodal body loads are applied with the **BF (p. 221)** command. Issue the **BFLIST (p. 255)** command to list the nodal body loads. The operations occur when the next body loads are defined. For example, issuing the **BF (p. 221)** command

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with a temperature of 250 after a previous **BF (p. 221)** command with a temperature of 200 causes the new value of the temperature to be 450 with the add operation, 250 with the replace operation, or 200 with the ignore operation. A scale factor is also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a temperature of 700. The scale factor is applied even if no previous values exist. Issue **BFCUM (p. 233)**,STAT to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note:

FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

BFCUM (p. 233) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Nodal Body Ld

Main Menu>Solution>Define Loads>Settings>Replace vs Add>Nodal Body Ld

---

BFDELE, *Node*, *Lab*

Deletes nodal body-force loads.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Node

Node at which body load is to be deleted. If ALL, delete for all selected nodes (**NSEL (p. 1266)**). If *Node* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You can substitute a component name for *Node*.

Lab

Valid body load label. If ALL, use all appropriate labels. See **BF (p. 221)**.

Notes

Deletes nodal body-force loads for a specified node and label. Nodal body loads are defined via **BF (p. 221)**.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Node Components

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Nodes

Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Nodes

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelMagField>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Node Components

Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Nodes

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFE, *Elem*, *Lab*, *STLOC*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Defines an element body-force load.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Elem

The element to which body load applies. If ALL, apply to all selected elements (**ESEL (p. 661)**). A component name may also be substituted for *Elem*.

Lab

Valid body load label. Valid labels are also listed for each element type in the *Element Reference* under "Body Loads" in the input table.

Discipline	Body Load Label	Label Description
Structural	TEMP	Temperature
	FREQ	Frequency (harmonic analyses only)
	FLUE	Fluence
	FORC	Body-force density
Thermal	HGEN	Heat generation rate (updated by volume changes when large-deflection effects are included (**NLGEOM (p. 1231)**,ON))
Magnetic	EF	Electric field
	TEMP	Temperature
	JS	Current density
	MVDI	Magnetic virtual displacements flag
Electric	TEMP	Temperature
	CHRGD	Charge density
Field volume interface	FVIN	Field volume interface flag
Poromechanics	FSOU	Fluid flow source
Diffusion	TEMP	Temperature
	DGEN	Diffusing substance generation rate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STLOC

Starting location for entering *VAL* data, below. For example, if *STLOC* = 1, data input in the *VAL1* field applies to the first element body load item available for the element type, *VAL2* applies to the second element item, etc. If *STLOC* = 5, data input in the *VAL1* field applies to the fifth element item, etc. Defaults to 1.

VAL1, VAL2, VAL3, VAL4

For *Lab* = TEMP, FLUE, DGEN, HGEN, and CHRGD, *VAL1*--*VAL4* represent body load values at the starting location and subsequent locations (usually nodes) in the element. *VAL1* can also represent a table name for use with tabular boundary conditions. Enter only *VAL1* for a uniform body load across the element. For nonuniform loads, the values must be input in the same order as shown in the input table for the element type. Values initially default to the **BFUNIF (p. 259)** value (except for CHRGD which defaults to zero). For subsequent specifications, a blank leaves a previously specified value unchanged; if the value was not previously specified, the default value as described in the *Element Reference* is used.

For *Lab* = JS and *STLOC* = 1, *VAL1*, *VAL2* and *VAL3* are the X, Y, and Z components of current density (in the element coordinate system), and *VAL4* is the phase angle.

For *Lab* = EF and *STLOC* = 1, *VAL1*, *VAL2*, and *VAL3* are the X, Y, and Z components of electric field (in the global Cartesian coordinate system).

If *Lab* = FVIN in a unidirectional Mechanical APDL to Ansys CFX analysis, *VAL2* is the volume interface number (not available from within the GUI), and *VAL1*, *VAL3*, and *VAL4* are not used.

For *Lab* = FORC and *STLOC* = 1, *VAL1*, *VAL2*, and *VAL3* are the real X, Y, and Z components of force density (in the global Cartesian coordinate system).

For analyses that allow complex input, if *Lab* = FORC and *STLOC* = 4, *VAL1*, *VAL2*, and *VAL3* are the imaginary X, Y, and Z components of force density (in the global Cartesian coordinate system).

Notes

Defines an element body-force load (such as the temperature in a structural analysis or the heat-generation rate in a thermal analysis). Body loads and element specific defaults are described for each element type in the *Element Reference*. If both the **BF (p. 221)** and **BFE (p. 237)** commands are used to apply a body-force load to an element, the **BFE (p. 237)** command takes precedence.

Body-Force Density (FORC)

Imaginary values for FORC loading via **BFE (p. 237)** is supported by current-technology solid elements (PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, and SOLID285) and reinforcing elements (REINF263, REINF264, and REINF265). Use only for modal or harmonic analyses. Large-deflection effects must be disabled (**NLGEOM (p. 1231)**,OFF).

Heat-Generation Loading (HGEN)

The following topics for applying HGEN loading via the **BFE (p. 237)** command are available:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On Layered Thermal Solid Elements

For HGEN loading on layered thermal solid elements SOLID278 / SOLID279 (KEYOPT(3) = 1 or 2), or layered thermal shell elements SHELL131 / SHELL132 (KEYOPT(3) = 1), *STLOC* refers to the *layer* number (not the node). In such cases, specify *VAL1* through *VAL4* to specify the heat-generation values for the appropriate layers. Heat generation is constant over the layer.

On Reinforcing Elements

For HGEN loading on reinforcing elements REINF263, REINF264, and REINF265, *STLOC* refers to the corner locations of the reinforcing members (individual reinforcings):

- REINF263 and REINF264: Specify *VAL1* and *VAL2* for each member. For tables, specify *VAL1* only.
- REINF265: Specify *VAL1*, *VAL2*, *VAL3*, and *VAL4* for each member. For tables, specify *VAL1* only.

For FORC loading on reinforcing elements, *STLOC* refers to real (*STLOC* = 1) or imaginary (*STLOC* = 4) components.

When using the standard method for defining reinforcing, this is the only way to apply a body load (HGEN or FORC) on the reinforcing members created after generating the REINF*nnn* reinforcing elements (**EREINF (p. 656)**). If applying FORC loading, Mechanical APDL applies a uniform load to all reinforcing members if there are multiple members in selected elements.

When using the mesh-independent method for defining reinforcing, you can apply a body load on the reinforcing members in the same way. The *preferred* method, however, is to apply loads on the MESH200 elements (via **BFE (p. 237)** or **BF (p. 221)** for HGEN, BFE for FORC) *before* generating the REINF*nnn* reinforcing elements (**EREINF (p. 656)**). Mechanical APDL maps the loads from the MESH200 elements to the newly generated REINF*nnn* reinforcing elements automatically. If you need to apply the loads *after* generating the reinforcing elements, apply them to MESH200 elements and issue **BFPORT (p. 257)** to transfer the loads to the reinforcing members.

Specifying a Table

You can specify a table name (*VAL1*) when using temperature (TEMP), diffusing substance generation rate (DGEN), heat generation rate (HGEN), and current density (JS) body load labels.

For the body-force-density label (FORC), you can specify a table for any of the *VAL1* through *VAL3* arguments. Both 1D and 2D tables are valid; however, only 1D tables are valid in mode-superposition harmonic and mode-superposition transient analyses.

Enclose the table name (*tabname*) in percent signs (%), for example:

BFE (p. 237),*Elem*, *Lab*,*STLOC*,%*tabname*%

Use the ***DIM (p. 530)** command to define a table. For information on primary variables for each load type, see Applying Loads Using Tabular Input in the *Basic Analysis Guide*.

For *Lab* = TEMP, each table defines *NTEMP* temperatures, as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For layered elements, *NTEMP* is the number of layer interface corners that allow temperature input.
- For non-layered elements, *NTEMP* is the number of corner nodes.

The temperatures apply to element items with a starting location of *STLOC* + *n*, where n is the value field location (*VALn*) of the table name input.

For layered elements, a single **BFE (p. 237)** command returns temperatures for one layer interface. Multiple **BFE (p. 237)** commands are necessary for defining all layered temperatures.

For beam, pipe and elbow elements that allow multiple temperature inputs per node, define the tabular load for the first node only (Node I), as loads on the remaining nodes are applied automatically. For example, to specify a tabular temperature load on a PIPE288 element with the through-wall-gradient option (KEYOPT(1) = 0), the **BFE (p. 237)** command looks like this:

BFE (p. 237),*Elem*,TEMP,1,%*tabOut*%, %*tabIn%*

where %*tabOut*% and %*tabIn*% and are the tables applied to the outer and inner surfaces of the pipe wall, respectively.

When a tabular function load is applied to an element, the load does not vary according to the positioning of the element in space.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

Other Information

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Volume Intr>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>Electric Field>On Elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Fluence>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Elements
Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Elements
Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Elements
Main Menu>Solution>Define Loads>Apply>Field Volume Intr>On Elements
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Elements
Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Normal Velo>On Elements
Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Elements
Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Elements
Main Menu>Solution>Define Loads>Apply>Magnetic>Other>Electric Field>On Elements
Main Menu>Solution>Define Loads>Apply>Structural>Fluence>On Elements
Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Elements

BFECUM, *Lab*, *Oper*, *FACT*, *TBASE*

Specifies whether to ignore subsequent element body force loads.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density
Diffusion	DGEN	diffusing substance generation rate

Oper

Replace or ignore key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REPL

Subsequent values replace the previous values (default).

IGNO

Subsequent values are ignored.

FACT

Scale factor for the element body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

TBASE

Used (only with *Lab* = TEMP) to calculate the temperature used in the add or replace operation (see *Oper*) as:

Temperature = *TBASE* + *FACT** (*T* - *TBASE*)

where *T* is the temperature specified on subsequent **BFE (p. 237)** commands. *TBASE* defaults to zero.

Command Default

Replace previous values.

Notes

Allows repeated element body-force loads to be replaced or ignored. Element body loads are applied with the **BFE (p. 237)** command. Issue the **BFELIST (p. 244)** command to list the element body loads. The operations occur when the next body loads are defined. For example, issuing the **BFE (p. 237)** command with a temperature value of 25 after a previous **BFE (p. 237)** command with a temperature value of 20 causes the new value of that temperature to be 25 with the replace operation, or 20 with the ignore operation. A scale factor is also available to multiply the next value before the replace operation. A scale factor of 2.0 with the previous "replace" example results in a temperature of 50. The scale factor is applied even if no previous values exist. Issue **BFECUM (p. 241)**,STAT to show the current label, operation, and scale factors.

BFECUM (p. 241) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Elem Body Lds

Main Menu>Solution>Define Loads>Settings>Replace vs Add>Elem Body Lds

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFEDELE, *ELEM*, *Lab*

Deletes element body-force loads.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element at which body load is to be deleted. If ALL, delete for all selected elements [A component name may also be substituted for *ELEM*.

Lab

Valid body load label. If ALL, use all appropriate labels. See **BFE (p. 237)** command for labels.

Notes

Deletes element body-force loads for a specified element and label. Element body loads may be defined with the **BFE (p. 237)** commands.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Elems

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Element Components

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Elements

Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Elems

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Elements
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Elements
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Normal Velo>On Elements
Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Elements
Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Elements
Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Elements
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Element Components
Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Elements
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Elements

---

BFELIST, *ELEM*, *Lab*

Lists the element body-force loads.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element at which body load is to be listed. If ALL (or blank), list for all selected elements (**ESEL (p. 661)**). If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

Lab

Valid body load label. If ALL (or blank), use all appropriate labels. See **BFE (p. 237)** command for labels.

Notes

Lists the element body-force loads for the specified element and label. Element body loads may be defined with the **BFE (p. 237)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Elements
Utility Menu>List>Loads>Body Loads>On Picked Elems

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFESCAL, *Lab*, *FACT*, *TBASE*

Scales element body-force loads.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density
Diffusion	DGEN	diffusing substance generation rate

FACT

Scale factor for the element body load values. Zero (or blank) defaults to 1.0. Use a small number for a "zero" scale factor. The scale factor is not applied to body load phase angles.

TBASE

Base temperature for temperature difference. Used only with *Lab* = TEMP. Scale factor is applied to the temperature difference (*T* - *TBASE*) and then added to *TBASE*. *T* is the current temperature.

Notes

Scales element body-force loads on the selected elements in the database. Issue the **BFELIST (p. 244)** command to list the element body loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled. (Note that such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.)

BFESCAL (p. 245) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Elem Body Lds

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Elem Body Lds

---

BFINT,*Fname1*,*Ext1*,--,*Fname2*,*Ext2*,--,*KPOS*,*Clab*,*KSHS*,*TOLOUT*,*TOLHGT*

Activates the body force interpolation operation.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname1

File name and directory path (248 characters maximum, including directory) from which to read data for interpolation. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext1

Filename extension (eight-character maximum).

The extension defaults to NODE if *Fname1* is blank.

--

Unused field.

Fname2

File name and directory path (248 characters maximum, including directory) to which **BF** commands are written. If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext2

Filename extension (eight-character maximum).

The extension defaults to BFIN if *Fname2* is blank.

--

Unused field.

KPOS

Position on *Fname2* to write block of **BF** commands:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Beginning of file (overwrite existing file).

1

End of file (append to existing file).

Clab

Label (8 characters maximum, including the colon) for this block of **BF** commands in *Fname2*. This label is appended to the colon (:). Defaults to BF*n*, where *n* is the cumulative iteration number for the data set currently in the database.

KSHS

Shell-to-solid submodeling key:

0

Solid-to-solid or shell-to-shell submodel.

1

Shell-to-solid submodel.

TOLOUT

Extrapolation tolerance about elements, based on a fraction of the element dimension. Submodel nodes outside the element by more than *TOLOUT* are not accepted as candidates for DOF extrapolation. Defaults to 0.5 (50%).

TOLHGT

Height tolerance above or below shell elements, in units of length. Used only for shell-to-shell submodeling (*KSHS* = 0). Submodel nodes off the element surface by more than *TOLHGT* are not accepted as candidates for DOF interpolation or extrapolation. Defaults to 0.0001 times the maximum element dimension.

Caution:

Relaxing this tolerance to allow submodel nodes to be found may cause poor submodel results.

Notes

File *Fname1* should contain a node list for which body forces are to be interpolated (**NWRITE (p. 1305)**). File *Fname2* is created, and contains interpolated body forces written as a block of nodal **BF (p. 221)** commands.

Body forces are interpolated from elements having TEMP as a valid body force or degree of freedom, and only the label TEMP is written on the nodal **BF (p. 221)** commands. Interpolation is performed for all nodes on file *Fname1* using the results data currently in the database. For layered elements, use the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LAYER (p. 1015) command to select the locations of the temperatures to be used for interpolation. Default locations are the bottom of the bottom layer and the top of the top layer.

The block of **BF (p. 221)** commands begins with an identifying colon label command and ends with a **/EOF (p. 645)** command. The colon label command has the form :*Clab*. Interpolation from multiple results sets can be performed by looping through the results file in a user-defined macro. Additional blocks can be appended to *Fname2* by using *KPOS* and unique colon labels. Issue **/INPUT (p. 948)**, with the appropriate colon label, to read the block of commands.

If the model has coincident (or very close) nodes, **BFINT (p. 246)** must be applied to each part of the model separately to ensure that the mapping of the nodes is correct. For example, if nodes belonging to two adjacent parts linked by springs are coincident, the operation should be performed on each part of the model separately.

Menu Paths

Main Menu>General Postproc>Submodeling>Interp Body Forc

BFK, *Kpoi*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *PHASE*

Defines a body-force load at a keypoint.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Kpoi

Keypoint to which body load applies. If ALL, apply to all selected keypoints (**KSEL (p. 995)**). A component name may also be substituted for *Kpoi*.

Lab

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Body Load Label	Label Description
Acoustic	JS	mass source or mass source rate
	TEMP	temperature

All keypoints on a given area (or volume) must have the same **BFK (p. 248)** table name for the tables to be transferred to interior nodes.

VAL1*, *VAL2*, *VAL3

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, MVDI and CHRGD. For magnetics, use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS. For acoustics, if *Lab* = JS, use *VAL1* for mass source in a harmonic analysis or mass source rate in a transient analysis, and ignore*VAL2* and *VAL3*. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFK (p. 248)**,*Kpoi*,*Lab*,%*tabname*%. Use the ***DIM (p. 530)** command to define a table.

PHASE

Phase angle in degrees associated with the JS label.

Notes

Defines a body-force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) at a keypoint. Body loads may be transferred from keypoints to nodes with the **BFTRAN (p. 259)** or **SBCTRAN (p. 1647)** commands. Interpolation will be used to apply loads to the nodes on the lines between keypoints. All keypoints on a given area (or volume) must have the same **BFK (p. 248)** specification, with the same values, for the loads to be transferred to interior nodes in the area (or volume). If only one keypoint on a line has a **BFK (p. 248)** specification, the other keypoint defaults to the value specified on the **BFUNIF (p. 259)** command.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFK (p. 248)** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Fluence>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Keypoints

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Keypoints

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Keypoints

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppVirtDisp>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Fluence>On Keypoints

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Keypoints

---

BFKDELE,*KPOI*,*Lab*

Deletes body-force loads at a keypoint.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

Keypoint at which body load is to be deleted. If ALL, delete for all selected keypoints (**KSEL (p. 995)**). A component name may also be substituted for *KPOI*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFK (p. 248)** command for labels.

Notes

Deletes body-force loads (and all corresponding finite element loads) for a specified keypoint and label. Body loads may be defined at a keypoint with the **BFK (p. 248)** command.

Graphical picking is available only via the listed menu paths.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All KPs

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Keypoints

Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All KPs

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Keypoints

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Keypoints

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Heat Generat>On Keypoints

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Keypoints

Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Keypoints

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppVirtDisp>On Keypoints

Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Keypoints

Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Keypoints

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Keypoints

---

BFKLIST, *KPOI*, *Lab*

Lists the body-force loads at keypoints.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

Keypoint at which body load is to be listed. If ALL (or blank), list for all selected keypoints (**KSEL (p. 995)**). If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFK (p. 248)** command for labels.

Notes

Lists the body-force loads for the specified keypoint and label. Keypoint body loads may be defined with the **BFK (p. 248)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Keypoints

Utility Menu>List>Loads>Body Loads>On Picked KPs

BFL, *Line*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*

Defines a body-force load on a line.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Line

Line to which body load applies. If ALL, apply to all selected lines (**LSEL (p. 1074)**). A component name may also be substituted for *Line*.

Lab

Valid body load label. Load labels are listed under "Body loads" in the input table for each element type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural Thermal	TEMP	temperature
	FLUE	fluence
	HGEN	heat generation rate
Magnetic	TEMP	temperature
Electric	TEMP	temperature
	CHRGD	charge density
Acoustic	JS	mass source or mass source rate
	TEMP	temperature

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

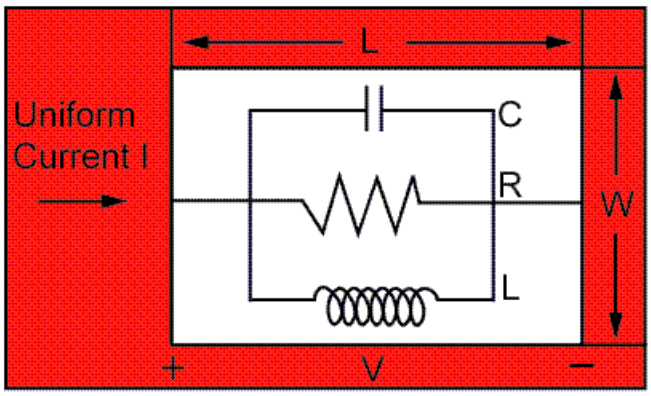
VAL1, VAL2, VAL3

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, and CHRGD. For acoustics, if *Lab* = JS, use *VAL1* for mass source in a harmonic analysis or mass source rate in a transient analysis, and ignore*VAL2* and *VAL3*. When specifying a table name, you must enclose the table name in percent signs (%), for example, **BFL (p. 252)**,*Line*,*Lab*,%*tabname*%. Use the ***DIM (p. 530)** command to define a table.

VAL4

If *Lab* = JS, *VAL4* is the phase angle in degrees.

Figure 1: RCL Lumped Circuit



Notes

Defines a body-force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on a line. Body loads may be transferred from lines to line elements (or to nodes if line elements do not exist) with the **BFTRAN (p. 259)** or **SBCTRAN (p. 1647)** commands.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFL (p. 252)** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppElecField>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Fluence>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppElecField>On Lines

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Lines

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Lines

Main Menu>Solution>Define Loads>Apply>Structural>Fluence>On Lines

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Lines

---

BFLDELE, *LINE*, *Lab*

Deletes body-force loads on a line.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

Line at which body load is to be deleted. If ALL, delete for all selected lines (**LSEL (p. 1074)**). A component name may also be substituted for *LINE*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFL (p. 252)** command for labels.

Notes

Deletes body-force loads (and all corresponding finite element loads) for a specified line and label. Body loads may be defined on a line with the **BFL (p. 252)** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>DelElecField>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Lines

Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Lines

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Lines

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Lines

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>DelElecField>On Lines

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Lines

Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Lines

Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Lines

Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Lines

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Lines

---

BFLIST, *Node*, *Lab*

Lists the body-force loads on nodes.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Node

Node at which body load is to be listed. If ALL (or blank), list for all selected nodes (**NSEL (p. 1266)**). If *Node* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You can substitute a component name for *Node*.

Lab

Valid body load label. If ALL (or blank), use all appropriate labels. (See **BF (p. 221)**.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Lists the body-force loads for the specified node and label. Nodal body loads are defined via **BF (p. 221)**.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Nodes

Utility Menu>List>Loads>Body Loads>On Picked Nodes

BFLLIST, *LINE*, *Lab*

Lists the body-force loads on a line.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

Line at which body load is to be listed. If ALL (or blank), list for all selected lines (**LSEL (p. 1074)**). If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFL (p. 252)** command for labels.

Notes

Lists the body-force loads for the specified line and label. Body loads may be defined on a line with the **BFL (p. 252)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Lines

Utility Menu>List>Loads>Body Loads>On Picked Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFPORT, *Cmname*

Transfers a thermal body-force load (HGEN) from selected MESH200 elements to reinforcing elements.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Cmname

Component name containing a list of selected MESH200 elements.

Notes

This command transfers a thermal body-force load (HGEN) from selected MESH200 elements to associated reinforcing elements or members (individual reinforcings). The association is established via **EREINF (p. 656)** using the mesh-independent method for defining reinforcing.

Issue this command *after* issuing **EREINF (p. 656)**.

Select MESH200 elements by issuing this command and specifying the component, or by issuing **ESEL (p. 661)**. (If you specify a component name, **ESEL (p. 661)** is ignored.)

This command supports a thermal body-force load (HGEN) only.

To define the thermal body-force load on MESH200 elements, issue **BFE (p. 237)** or **BF (p. 221)**.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

BFSCALE, *Lab*, *FACT*, *TBASE*

Scales body-force loads at nodes.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Body Load Label	Label Description
Magnetic	TEMP	temperature
	JS	current densities
	MVDI	magnetic virtual displacements flag
Electric	TEMP	temperature
	CHRGD	charge density
Diffusion	DGEN	diffusing substance generation rate

FACT

Scale factor for the nodal body load values. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor. The scale factor is not applied to body load phase angles.

TBASE

Base temperature for temperature difference. Used only with *Lab* = TEMP. Scale factor is applied to the temperature difference (*T* - *TBASE*) and then added to *TBASE*. *T* is the current temperature.

Notes

Scales body-force loads in the database on the selected nodes. Issue the **BFLIST (p. 255)** command to list the nodal body loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note:

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

BFSCALE (p. 257) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Nodal Body Ld

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Nodal Body Ld

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFTRAN

Transfers solid model body-force loads to the finite element model.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Body loads are transferred from selected keypoints and lines to selected nodes and from selected areas and volumes to selected elements. The **BFTRAN (p. 259)** operation is also done if the **SBCTRAN (p. 1647)** command is either explicitly issued or automatically issued upon initiation of the solution calculations (**SOLVE (p. 1822)**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Body Loads

Main Menu>Solution>Define Loads>Operate>Transfer to FE>Body Loads

BFUNIF, *Lab*, *VALUE*

Assigns a uniform body-force load to all nodes.

SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid body load label. If ALL, use all appropriate labels.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
	TEMP	initial temperature
Diffusion	DGEN	diffusing substance generation rate

VALUE

Uniform value associated with *Lab* item, or table name when specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), for example, **BFUNIF (p. 259)**,*Lab*,%*tabname*%.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Set TEMP to the reference temperature (**TREF (p. 2013)** but not **MP (p. 1160)**,REFT), and FLUE and HGEN to zero.

Notes

In a transient or nonlinear thermal analysis, the uniform temperature is used during the first iteration of a solution as follows: (a) as the starting nodal temperature except where temperatures are explicitly specified (**D (p. 483)**, **DK (p. 541)**), and (b) to evaluate temperature-dependent material properties. In a structural analysis, the uniform temperature is used as the *default* temperature for thermal strain calculations and material property evaluation except where body load temperatures are specified (**BF (p. 221)**, **BFE (p. 237)**, **BFK (p. 248)**, **LDREAD (p. 1036)**). In other scalar field analyses, the uniform temperature is used for material property evaluation.

An alternate command, **TUNIF (p. 2027)**, may be used to set the uniform temperature instead of **BFUNIF (p. 259)**,TEMP. Since **TUNIF (p. 2027)** (or **BFUNIF (p. 259)**,TEMP) is step-applied in the first iteration, you should use **BF (p. 221)**, ALL, TEMP, Value to ramp on a uniform temperature load.

You can specify a table name only when using temperature (TEMP), heat generation rate (HGEN), and diffusing substance generation rate (DGEN) body load labels. When using TEMP, you can define a one-dimensional table that varies with respect to time (TIME) only. When defining this table, enter TIME as the primary variable. No other primary variables are valid.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>Uniform Heat Gen

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Fluence>Uniform Fluen

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>Uniform Heat Gen

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>Uniform Heat Gen

Main Menu>Solution>Define Loads>Apply>Structural>Fluence>Uniform Fluen

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>Uniform Heat Gen

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFV, *Volu*, *Lab*, *VAL1*, *VAL2*, *VAL3*, *PHASE*

Defines a body-force load on a volume.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Volu

Volume to which body load applies. If ALL, apply to all selected volumes (**VSEL (p. 2163)**). A component name may also be substituted for *Volu*.

Lab

Valid body load label. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	TEMP	temperature
	FLUE	fluence
Thermal	HGEN	heat generation rate
Magnetic	TEMP	temperature
	JS	current densities
Electric	TEMP	temperature
	CHRGD	charge density
Acoustic	JS	mass source or mass source rate
	TEMP	temperature

VAL1, VAL2, VAL3

Value associated with the *Lab* item or a table name for specifying tabular boundary conditions. Use only *VAL1* for TEMP, FLUE, HGEN, and CHRGD. Use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components of JS. For *Lab* = JS in magnetics, use *VAL1*, *VAL2*, and *VAL3* for the X, Y, and Z components. For acoustics, if *Lab* = JS, use *VAL1* for mass source in a harmonic analysis or mass source rate in a transient analysis, and ignore*VAL2* and *VAL3*. When specifying a table name, you must enclose the table name in percent signs (%), e.g., **BFV (p. 261)**,*Volu*,*Lab*,%*tabname*%. Use the ***DIM (p. 530)** command to define a table.

PHASE

Phase angle in degrees associated with the JS label.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines a body-force load (such as temperature in a structural analysis, heat generation rate in a thermal analysis, etc.) on a volume. Body loads may be transferred from volumes to volume elements (or to nodes if volume elements do not exist) with the **BFTRAN (p. 259)** or **SBCTRAN (p. 1647)** commands. Body loads default to the value specified on the **BFUNIF (p. 259)** command, if it was previously specified.

You can specify a table name only when using temperature (TEMP) and heat generation rate (HGEN) body load labels.

Body loads specified by the **BFV (p. 261)** command can conflict with other specified body loads. See Resolution of Conflicting Body Load Specifications in the *Basic Analysis Guide* for details.

Graphical picking is available only via the listed menu paths.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharDens>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Fluence>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>On Volumes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>On Volumes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharDens>On Volumes

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>On Volumes

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>On Volumes

Main Menu>Solution>Define Loads>Apply>Structural>Fluence>On Volumes

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>On Volumes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BFVDELE, *VOLU*, *Lab*

Deletes body-force loads on a volume.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VOLU

Volume at which body load is to be deleted. If ALL, delete for all selected volumes (**VSEL (p. 2163)**). A component name may also be substituted for *VOLU*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFV (p. 261)** command for labels.

Notes

Deletes body-force loads (and all corresponding finite element loads) for a specified volume and label. Body loads may be defined on a volume with the **BFV (p. 261)** command.

Graphical picking is available only via the listed menu paths.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Body Loads>On All Volms

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Temperature>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharDens>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>Temperature>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Other>Fluence>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Temperature>On Volumes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Generat>On Volumes

Main Menu>Solution>Define Loads>Delete>All Load Data>All Body Loads>On All Volms

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Temperature>On Volumes

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharDens>On Volumes

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>Temperature>On Volumes

Main Menu>Solution>Define Loads>Delete>Magnetic>Excitation>DelCurrDens>On Volumes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Delete>Structural>Other>Fluence>On Volumes

Main Menu>Solution>Define Loads>Delete>Structural>Temperature>On Volumes

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Generat>On Volumes

---

BFVLIST, *VOLU*, *Lab*

Lists the body-force loads on a volume.

SOLUTION (p. 38): Solid Body Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VOLU

Volume at which body load is to be listed. If ALL (or blank), list for all selected volumes (**VSEL (p. 2163)**). If *VOLU* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *VOLU*.

Lab

Valid body load label. If ALL, use all appropriate labels. Load labels are listed under "Body Loads" in the input table for each element type in the *Element Reference*. See the **BFV (p. 261)** command for labels.

Notes

Lists the body-force loads for the specified volume and label. Body loads may be defined on a volume with the **BFV (p. 261)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Body Loads>On All Volumes

Utility Menu>List>Loads>Body Loads>On Picked Volumes

BIOOPT

Specifies "Biot-Savart options" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Biot Savart Options

BIOT, *Label*

Calculates the Biot-Savart source magnetic field intensity.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Controls the Biot-Savart calculation:

NEW

Calculate the magnetic source field intensity (H_s) from the selected set of source elements to the selected set of nodes. Overwrite any existing H_s field values.

SUM

Calculate the H_s field from the selected set of source elements to the selected set of nodes. Accumulate with any existing H_s field values.

Command Default

Calculate the H_s field upon encountering the first **SOLVE (p. 1822)** command to produce a source field.

Notes

Calculates the Biot-Savart source magnetic field intensity (H_s) at the selected nodes from the selected source elements. The calculation is done at the time the **BIOT (p. 265)** command is issued.

Source elements include primitives described by element SOURC36, and coupled-field elements SOLID5, LINK68, and SOLID98. Current conduction elements do not have a solved-for current distribution from which to calculate a source field until after the first substep. Inclusion of a current conduction element H_s field will require a subsequent **BIOT (p. 265)**,SUM command (with SOURC36 elements unselected) and a **SOLVE (p. 1822)** command.

The units of H_s are as specified by the current **EMUNIT (p. 632)** command setting.

This command is also valid in PREP7.

Distributed-Memory Parallel (DMP) Restriction

When used with SOLID5, LINK68, or SOLID98, the **BIOT (p. 265)** command is not supported in a DMP solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Biot-Savart

Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Biot-Savart

---

BLC4,*XCORNER*,*YCORNER*,*WIDTH*,*HEIGHT*,*DEPTH*

Creates a rectangular area or block volume by corner points.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCORNER, YCORNER

Working plane X and Y coordinates of one corner of the rectangle or block face.

WIDTH

The distance from *XCORNER* on or parallel to the working plane X-axis that, together with *YCORNER*, defines a second corner of the rectangle or block face.

HEIGHT

The distance from *YCORNER* on or parallel to the working plane Y-axis that, together with *XCORNER*, defines a third corner of the rectangle or block face.

DEPTH

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the block. If *DEPTH* = 0 (default), a rectangular area is created on the working plane.

Notes

Defines a rectangular area anywhere on the working plane or a hexahedral volume with one face anywhere on the working plane. A rectangle will be defined with four keypoints and four lines. A volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC5 (p. 267)**, **RECTNG (p. 1580)**, and **BLOCK (p. 268)** commands for alternate ways to create rectangles and blocks.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By 2 Corners

Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By 2 Corners & Z

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BLC5,*XCENTER*,*YCENTER*,*WIDTH*,*HEIGHT*,*DEPTH*

Creates a rectangular area or block volume by center and corner points.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCENTER, YCENTER

Working plane X and Y coordinates of the center of the rectangle or block face.

WIDTH

The total distance on or parallel to the working plane X-axis defining the width of the rectangle or block face.

HEIGHT

The total distance on or parallel to the working plane Y-axis defining the height of the rectangle or block face.

DEPTH

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the block. If *DEPTH* = 0 (default), a rectangular area is created on the working plane.

Note:

If you are working with a model imported from an IGES file (import option set to DEFAULT), you must supply a value for *DEPTH* or the command is ignored.

Notes

Defines a rectangular area anywhere on the working plane or a hexahedral volume with one face anywhere on the working plane by specifying the center and corner points. A rectangle will be defined with four keypoints and four lines. A volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC4 (p. 266)**, **RECTNG (p. 1580)**, and **BLOCK (p. 268)** commands for alternate ways to create rectangles and blocks.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By Centr & Cornr

Main Menu>Preprocessor>Modeling>Create>Primitives>Block

Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By Centr,Cornr,Z

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BLOCK,*X1*,*X2*,*Y1*,*Y2*,*Z1*,*Z2*

Creates a block volume based on working plane coordinates.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X1, X2

Working plane X coordinates of the block.

Y1, Y2

Working plane Y coordinates of the block.

Z1, Z2

Working plane Z coordinates of the block.

Notes

Defines a hexahedral volume based on the working plane. The block must have a spatial volume greater than zero (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The volume will be defined with eight keypoints, twelve lines, and six areas, with the top and bottom faces parallel to the working plane. See the **BLC4 (p. 266)** and **BLC5 (p. 267)** commands for alternate ways to create blocks.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Block>By Dimensions

BOOL

Specifies "Booleans" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>List>Status>Preprocessor>Solid Model

BOPTN, *Lab*, *Value*

Specifies Boolean operation options.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Default/status key:

DEFA

Resets settings to default values.

STAT

Lists status of present settings.

Option to be controlled:

KEEP

Delete or keep input entity option.

NUMB

Output numbering warning message option.

NWARN

No effect warning message option.

VERSION

Boolean compatibility option.

Value

Option settings if *Lab* = KEEP:

NO

Delete entities used as input with a Boolean operation (default). Entities will not be deleted if meshed or if attached to a higher entity.

YES

Keep input solid modeling entities.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option settings if *Lab* = NUMB:

0

No warning message will be produced if the output entities of a Boolean operation are numbered based on geometry (default).

1

A warning message will be produced if the output entities of a Boolean operation are numbered based on geometry. (With geometric numbering, re-use of the input with altered dimensions may not produce the same numbering, and later operations in the input may fail or produce unexpected results.)

Option settings if *Lab* = NWARN:

0

A warning message will be produced if a Boolean operation has no effect (default).

1

No warning or error messages will be generated if a Boolean operation has no effect.

-1

An error message will be produced if a Boolean operation has no effect.

Option settings if *Lab* = VERSION:

RV52

Activate the Revision 5.2 compatibility option (default). The 5.2 option can produce different numbering of the entities produced by Boolean operations than the 5.1 option. See Notes below.

RV51

Activate the Revision 5.1 compatibility option. The 5.1 option can produce different numbering of the entities produced by Boolean operations than the 5.2 option. See "Notes" (p. 270) below.

Command Default

Input entities will be deleted, and operations with no effect (that is, operations which are valid but which do not cause a change in the input entities, such as adding two non-touching areas) will produce a warning message. The Revision 5.2 Boolean compatibility option will be used.

Notes

Boolean operations at Revision 5.2 may produce a different number of entities than previous revisions of Mechanical APDL. When running input files created in earlier versions of Mechanical APDL, match the Boolean compatibility option (VERSION) to the revision originally used. For instance, if you are running

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Revision 5.2 and are reading an input file (**/INPUT (p. 948)**) created at Revision 5.1, it is recommended that you set VERSION to RV51 before reading the input.

See the *Modeling and Meshing Guide* for further details on the functions of the RV51 and RV52 labels.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Settings

BSAX, *VAL1*, *VAL2*, *T*

Specifies the axial strain and axial force relationship for beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1

Axial strain component (ε).

VAL2

Axial force component (N).

T

Temperature.

Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon,T) & & & & & \\ & I_1E(\kappa_1,T) & & 0 & & \\ & & I_2E(\kappa_2,T) & & & \\ & & & JG(\chi,T) & & \\ & 0 & & & A_1G(\gamma_1,T) & \\ & & & & & A_2G(\gamma_2,T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSAX (p. 271)** command, one of several nonlinear general beam section commands, specifies the relationship of axial strain and axial force for a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero, with the exception of temperature. If the temperature is unspecified, the values become temperature-independent.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related commands are **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, and **BSTE (p. 278)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

BSMD, *DENS*, *T*

Specifies mass per unit length for a nonlinear general beam section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DENS

Mass density.

T

Temperature.

Notes

The **BSMD (p. 272)** command, one of several nonlinear general beam section commands, specifies the mass density (assuming a unit area) for a beam section. The value specified is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

If *DENS* is unspecified, it defaults to zero. If the temperature is unspecified, *DENS* becomes temperature-independent.

Related commands are **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, and **BSTE (p. 278)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BSM1, *VAL1*, *VAL2*, *T*

Specifies the bending curvature and moment relationship in plane XZ for beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1

Curvature component (κ_1).

VAL2

Bending moment component (M_1).

T

Temperature.

Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon,T) & & & & & \\ & I_1E(\kappa_1,T) & & & 0 & \\ & & I_2E(\kappa_2,T) & & & \\ & & & JG(\chi,T) & & \\ & 0 & & & A_1G(\gamma_1,T) & \\ & & & & & A_2G(\gamma_2,T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSM1 (p. 273)** command, one of several nonlinear general beam section commands, specifies the bending curvature and moment for plane XZ of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero, with the exception of temperature. If the temperature is unspecified, the values become temperature-independent.

Related commands are **BSAX (p. 271)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, and **BSTE (p. 278)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide* .

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BSM2, *VAL1*, *VAL2*, *T*

Specifies the bending curvature and moment relationship in plane XY for beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1

Curvature component (κ_2).

VAL2

Bending moment component (M_2).

T

Temperature.

Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon,T) & & & & & \\ & I_1E(\kappa_1,T) & & & 0 & \\ & & I_2E(\kappa_2,T) & & & \\ & & & JG(\chi,T) & & \\ & 0 & & & A_1G(\gamma_1,T) & \\ & & & & & A_2G(\gamma_2,T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSM2 (p. 274)** command, one of several nonlinear general beam section commands, specifies the bending curvature and moment relationship for plane XY of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero, with the exception of temperature. If the temperature is unspecified, the values become temperature-independent.

Related commands are **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, and **BSTE (p. 278)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide* .

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BSPLIN, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *XV1*, *YV1*, *ZV1*, *XV6*, *YV6*, *ZV6*

Generates a single line from a spline fit to a series of keypoints.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1, P2, P3, . . . , P6

Keypoints through which a spline is fit. At least two keypoints must be defined. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

The following fields are used only if specified end slopes are desired; otherwise, zero curvature end slopes will be automatically calculated. The slope vector is parallel to a vector pointing from the origin of the active coordinate system (**CSYS (p. 441)**) to the position in space that XV, YV, ZV represents in that system.

XV1, YV1, ZV1

Orientation point of an outward vector tangent to line at *P1*. Vector coordinate system has its origin at the keypoint. Coordinate interpretation corresponds to the active coordinate system type, that is, X is R for cylindrical, etc. Defaults to zero curvature slope.

XV6, YV6, ZV6

Orientation point of an outward vector tangent to a line at *P6* (or the last keypoint specified if fewer than six specified). Defaults to zero curvature slope.

Notes

One line is generated between keypoint *P1* and the last keypoint entered. The line will pass through each entered keypoint. Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Spline thru KPs

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Spline thru Locs

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Spline thru KPs

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Spline thru Locs

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BSS1, *VAL1*, *VAL2*, *T*

Specifies the transverse shear strain and force relationship in plane XZ for beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1

Transverse shear strain component (γ_1).

VAL2

Transverse shear force component (S_1).

T

Temperature.

Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon,T) & & & & & \\ & I_1E(\kappa_1,T) & & & 0 & \\ & & I_2E(\kappa_2,T) & & & \\ & & & JG(\chi,T) & & \\ & 0 & & & A_1G(\gamma_1,T) & \\ & & & & & A_2G(\gamma_2,T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSS1 (p. 276)** command, one of several nonlinear general beam section commands, specifies the transverse shear strain and transverse shear force relationship for plane XZ of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero, with the exception of temperature. If the temperature is unspecified, the values become temperature-independent.

Related commands are **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, and **BSTE (p. 278)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide*
.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BSS2, *VAL1*, *VAL2*, *T*

Specifies the transverse shear strain and force relationship in plane XY for beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1

Transverse shear strain component (γ_2).

VAL2

Transverse shear force component (S_2).

T

Temperature.

Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon,T) & & & & & \\ & I_1E(\kappa_1,T) & & & 0 & \\ & & I_2E(\kappa_2,T) & & & \\ & & & JG(\chi,T) & & \\ & 0 & & & A_1G(\gamma_1,T) & \\ & & & & & A_2G(\gamma_2,T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSS1 (p. 276)** command, one of several nonlinear general beam section commands, specifies the transverse shear strain and transverse shear force relationship for plane XY of a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero, with the exception of temperature. If the temperature is unspecified, the values become temperature-independent.

Related commands are **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSMD (p. 272)**, and **BSTE (p. 278)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide*
.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BSTE, *ALPHA*, *T*

Specifies a thermal expansion coefficient for a nonlinear general beam section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ALPHA

Coefficient of thermal expansion for the cross section.

T

Temperature.

Notes

The **BSTE (p. 278)** command, one of several nonlinear general beam section commands, specifies a thermal expansion coefficient for a beam section. The value specified is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

If *ALPHA* is unspecified, it defaults to zero. If the temperature is unspecified, *ALPHA* becomes temperature-independent.

Related commands are **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, and **BSMD (p. 272)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

BSTQ, *VAL1*, *VAL2*, *T*

Specifies the cross section twist and torque relationship for beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1

Twist component (χ).

VAL2

Torque component (τ).

T

Temperature.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \end{bmatrix} = \begin{bmatrix} AE(\varepsilon,T) & & & & & \\ & I_1E(\kappa_1,T) & & 0 & & \\ & & I_2E(\kappa_2,T) & & & \\ & & & JG(\chi,T) & & \\ & 0 & & & A_1G(\gamma_1,T) & \\ & & & & & A_2G(\gamma_2,T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \end{bmatrix}$$

The **BSTQ (p. 278)** command, one of several nonlinear general beam section commands, specifies the cross section twist and torque relationship for a beam section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero, with the exception of temperature. If the temperature is unspecified, the values become temperature-independent.

Related commands are **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, and **BSTE (p. 278)**.

For complete information, see Using Nonlinear General Beam Sections in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>NL Generalized

BTOL, *PTOL*

Specifies the Boolean operation tolerances.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

PTOL

Point coincidence tolerance. Points within this distance to each other will be assumed to be coincident during Boolean operations. Loosening the tolerance will increase the run time and storage requirements, but will allow more Boolean intersections to succeed. Defaults to 0.10E-4.

Command Default

PTOL = 0.10E-4.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Use **BTOL (p. 279)**,DEFA to reset the setting to its default value. Use **BTOL (p. 279)**,STAT to list the status of the present setting.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Settings

BUCOPT,*Method*,*NMODE*,*SHIFT*,*LDMULTE*,*RangeKey*

Specifies buckling analysis options.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Method

Mode extraction method to be used for the buckling analysis:

LANB

Block Lanczos

SUBSP

Subspace iteration

See Eigenvalue and Eigenvector Extraction in the *Mechanical APDL Theory Reference* for more information on these two methods.

NMODE

Number of buckling modes (that is, eigenvalues or load multipliers) to extract (defaults to 1).

SHIFT

By default, this value acts as the initial shift point about which the buckling modes are calculated (defaults to 0.0).

When *RangeKey* = RANGE, this value acts as the lower bound of the load multiplier range of interest (*LDMULTE* is the upper bound).

LDMULTE

Boundary for the load multiplier range of interest (defaults to ∞).

When *RangeKey* = CENTER, the *LDMULTE* value determines the lower and upper bounds of the load multiplier range of interest (-*LDMULTE*, +*LDMULTE*).

When *RangeKey* = RANGE, the *LDMULTE* value is the upper bound for the load multiplier range of interest (*SHIFT* is the lower bound).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RangeKey

Key used to control the behavior of the eigenvalue extraction method (defaults to CENTER):

CENTER

Use the CENTER option control (default); the program computes *NMODE* buckling modes centered around *SHIFT* in the range of (-*LDMULTE*, +*LDMULTE*).

RANGE

Use the RANGE option control; the program computes *NMODE* buckling modes in the range of (*SHIFT*, *LDMULTE*).

Notes

Specifies buckling analysis (**ANTYPE (p. 162)**,BUCKLE) options. Additional options used only for the Block Lanczos (LANB) eigensolver are specified by the **LANBOPTION (p. 1007)** command. For more difficult buckling problems, you can specify an alternative version of the Block Lanczos eigensolver (**LANBOPTION (p. 1007)**,,,ALT1).

Eigenvalues from a buckling analysis can be negative and/or positive. The program sorts the eigenvalues from the most negative to the most positive values. The minimum buckling load factor may correspond to the smallest eigenvalue in absolute value, or to an eigenvalue within the range, depending on your application (that is, linear perturbation buckling analysis or purely linear buckling analysis).

It is recommended that you request an additional few buckling modes beyond what is needed in order to enhance the accuracy of the final solution. It is also recommended that you input a non zero *SHIFT* value and a reasonable *LDMULTE* value (that is, a smaller *LDMULTE* that is closer to the last buckling mode of interest) when numerical problems are encountered.

When using the RANGE option, defining a range that spans zero is not recommended. If you are seeking both negative and positive eigenvalues, it is recommended that you use the CENTER option.

This command is also valid in PREP7. If used in SOLUTION, this command is valid only within the first load step.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

C Commands

C***,*Comment*

Places a comment in the output.

SESSION (p. 11): List Controls (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Comment

Comment string, up to 75 characters.

Notes

The output from this command consists of two lines -- a blank line followed by a line containing C*** and the comment. This command is similar to **/COM (p. 404)** except that the comment produced by **C*** (p. 283)** is more easily identified in the output.

Another way to include a comment is to precede it with a ! character (on the same line). The ! may be placed anywhere on the line, and any input following it is ignored as a comment. No output is produced by such a comment, but the comment line is included on the log file. This is a convenient way to annotate the log file.

This command is valid anywhere.

Menu Paths

This command cannot be accessed from a menu.

CALC

Specifies "Calculation settings" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Calculations

CAMPBELL,*Action*

Prepares the result file for a subsequent Campbell diagram of a prestressed structure.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Campbell action:

NONE

Do not prepare the result file. This option is the default behavior.

RSTP

Prepare the result file (`Jobname.rstp`) for a subsequent Campbell diagram of a prestressed structure.

Notes

For an analysis involving a prestressed structure, the **CAMPBELL (p. 284)** command specifies whether or not to prepare the result file to support a Campbell diagram analysis (**PRCAMP (p. 1469)** or **PLCAMP (p. 1392)**).

To prestress a structure, the program performs a static solution before the linear perturbation modal solution. For specific information about rotating structures, see Considerations for Rotating Structures.

The **CAMPBELL (p. 284)** command requires that modal and static analyses be performed alternately. It works only when the number of static analyses is the same as the number of modal analyses. Any number of analyses can be performed, but the same number of each (static and modal) is expected. The modal solutions are appended in the results file (`Jobname.rstp`).

For an example of **CAMPBELL (p. 284)** command usage, see Example 5.1: Commands used in a Campbell Diagram Analysis of a Prestressed Structure in the *Rotordynamic Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CBDOF,*Fname1*,*Ext1*,--,*Fname2*,*Ext2*,--,*KPOS*,*Clab*,*KSHS*,*TOLOUT*,*TOLHGT*,*TOLTHK*

Activates cut-boundary interpolation (for submodeling).

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname1

File name and directory path (248 characters maximum, including directory) from which to read boundary node data. If no specified directory path exists, the path defaults to your working directory and you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext1

Filename extension (eight-character maximum).

The extension defaults to NODE if *Fname1* is blank.

--

Unused field.

Fname2

File name and directory path (248 characters maximum, including directory) to which cut-boundary **D** commands are written. If no specified directory path exists, the path defaults to your working directory and you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext2

Filename extension (eight-character maximum).

The extension defaults to CBDO if *Fname2* is blank.

--

Unused field.

KPOS

Position on *Fname2* to write block of **D (p. 483)** commands:

0

Beginning of file (overwrite existing file).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

End of file (append to existing file).

Clab

Label (eight characters maximum, including the colon) for this block of **D (p. 483)** commands on *Fname2*. his label is appended to the colon (:). Defaults to CB*n*, where *n* is the cumulative iteration number for the data set currently in the database. For imaginary data (see *KIMG* on the ***SET (p. 1720)** command), *Clab* defaults to CI*n*.

KSHS

Shell-to-solid submodeling key:

0

Solid-to-solid or shell-to-shell submodel.

1

Shell-to-solid submodel.

TOLOUT

Extrapolation tolerance about elements, based on a fraction of the element dimension. Submodel nodes outside the element by more than *TOLOUT* are not accepted as candidates for DOF extrapolation. Defaults to 0.5 (50 percent).

TOLHGT

Height tolerance above or below shell elements, in units of length. Used only for shell-to-shell submodeling (*KSHS* = 0). Submodel nodes off the element surface by more than *TOLHGT* are not accepted as candidates for degree-of-freedom interpolation or extrapolation. Defaults to 0.0001 times the maximum element dimension.

TOLTHK

Height tolerance above or below shell elements, based on a fraction of the shell element thickness. Used only for shell-to-solid submodeling (KSHS = 1). Submodel nodes off the element surface by more than *TOLTHK* are not accepted as candidates for DOF interpolation or extrapolation. Defaults to 0.1 times the average shell thickness.

Notes

File *Fname1* should contain a node list for which boundary conditions are to be interpolated (**NWRITE (p. 1305)**). File *Fname2* is created to contain interpolated boundary conditions written as a block of **D (p. 483)** commands.

Boundary conditions are written for the active degree-of-freedom set for the element from which interpolation is performed. Interpolation occurs on the selected set of elements. The block of **D (p. 483)** commands begins with an identifying colon label and ends with a **/EOF (p. 645)** command. The colon label is of the form :*Clab* (described above).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interpolation from multiple results sets can be performed by looping through the results file in a user-defined macro. Additional blocks can be appended to *Fname2* by using *KPOS* and unique colon labels. To read the block of commands, issue the **/INPUT (p. 948)** command with the appropriate colon label.

If the model has coincident (or very close) nodes, the **CBDOF (p. 285)** must be applied to each part of the model separately to ensure that the mapping of the nodes is correct. For example, if nodes belonging to two adjacent parts linked by springs are coincident, the operation should be performed on each part of the model separately.

Resume the coarse model database at the beginning of the cut-boundary procedure. The database should have been saved after the first coarse model solution, as the number of nodes in the database and the results file must match, and internal nodes are sometimes created during the solution.

This command cannot be used to interpolate the magnetic edge-flux (AZ) degree of freedom.

Caution:

Relaxing the *TOLHGT* or *TOLTHK* tolerances to allow submodel nodes to be "found" can produce poor submodel results.

Menu Paths

Main Menu>General Postproc>Submodeling>Interpolate DOF

CBMD, *ROW*, $C_{(R)(R)}$, $C_{(R)(R+1)}$, $C_{(R)(R+2)}$, $C_{(R)(R+3)}$, $C_{(R)(R+4)}$, $C_{(R)(R+5)}$

Specifies preintegrated section mass matrix for composite-beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ROW

Row number of the matrix.

$C_{(R)(R)}$, ..., $C_{(R)(R+5)}$

Upper triangle of the cross-section mass matrix **[C]**.

Notes

With a unit beam length, the section mass matrix relates the resultant forces and torques to accelerations and angular accelerations as follows (applicable to the local element coordinate system):

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} N_x \\ N_y \\ N_z \\ M_x \\ M_y \\ M_z \end{bmatrix} = \begin{bmatrix} C_{11}(T) & C_{12}(T) & C_{13}(T) & C_{14}(T) & C_{15}(T) & C_{16}(T) \\ & C_{22}(T) & C_{23}(T) & C_{24}(T) & C_{25}(T) & C_{26}(T) \\ & & C_{33}(T) & C_{34}(T) & C_{35}(T) & C_{36}(T) \\ & & & C_{44}(T) & C_{45}(T) & C_{46}(T) \\ & & & & C_{55}(T) & C_{56}(T) \\ & & & & & C_{66}(T) \end{bmatrix} \begin{bmatrix} A_x \\ A_y \\ A_z \\ R_x \\ R_y \\ R_z \end{bmatrix}$$

The **CBMD (p. 287)** command, one of several composite beam section commands, specifies the section mass matrix (submatrix [***C***] data) for a composite beam section. The section data defined is associated with the section most recently defined (**SECTYPE (p. 1697)**) at the specified temperature (**CBTMP (p. 290)**).

Unspecified values default to zero.

Related commands are **CBTMP (p. 290)**, **CBTE (p. 289)**, and **CBMX (p. 288)**.

For complete information, see Using Preintegrated Composite Beam Sections.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Composite Sections

CBMX, *ROW*, $S_{(R)(R)}$, $S_{(R)(R+1)}$, $S_{(R)(R+2)}$, $S_{(R)(R+3)}$, $S_{(R)(R+4)}$, $S_{(R)(R+5)}$, $S_{(R)(R+6)}$

Specifies preintegrated cross-section stiffness for composite beam sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ROW

Row number of the matrix.

$S_{(R)(R)}$, . . . , $S_{(R)(R+6)}$

Upper triangle of the cross-section stiffness matrix **[S]**.

Notes

The behavior of beam elements is governed by the generalized-stress/generalized-strain relationship of the form:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{bmatrix} N \\ M_1 \\ M_2 \\ \tau \\ S_1 \\ S_2 \\ B_m \end{bmatrix} = \begin{bmatrix} S_{11}(T) & S_{12}(T) & S_{13}(T) & S_{14}(T) & S_{15}(T) & S_{16}(T) & S_{17}(T) \\ & S_{22}(T) & S_{23}(T) & S_{24}(T) & S_{25}(T) & S_{26}(T) & S_{27}(T) \\ & & S_{33}(T) & S_{34}(T) & S_{35}(T) & S_{36}(T) & S_{37}(T) \\ & & & S_{44}(T) & S_{45}(T) & S_{46}(T) & S_{47}(T) \\ & & & & S_{55}(T) & S_{56}(T) & S_{57}(T) \\ & & & & & S_{66}(T) & S_{67}(T) \\ & & & & & & S_{77}(T) \end{bmatrix} \begin{bmatrix} \varepsilon \\ \kappa_1 \\ \kappa_2 \\ \chi \\ \gamma_1 \\ \gamma_2 \\ B_k \end{bmatrix}$$

Notes

The **CBMX (p. 288)** command, one of several composite beam section commands, specifies the cross-section stiffness matrix (submatrix [***S***] data) for a composite beam section. The section data defined is associated with the section most recently defined (**SECTYPE (p. 1697)**) at the specified temperature (**CBTMP (p. 290)**).

Unspecified values default to zero.

Related commands are **CBTMP (p. 290)**, **CBTE (p. 289)**, and **CBMD (p. 287)**.

For complete information, see Using Preintegrated Composite Beam Sections.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Composite Sections

CBTE, *ALPHA*

Specifies a thermal expansion coefficient for a composite beam section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ALPHA

Coefficient of thermal expansion for the cross section.

Notes

The **CBTE (p. 289)** command, one of several composite beam section commands, specifies a thermal expansion coefficient for a beam section. The value specified is associated with the section most recently defined (**SECTYPE (p. 1697)**) at the specified temperature (**CBTMP (p. 290)**).

Unspecified values default to zero.

Related commands are **CBTMP (p. 290)**, **CBMX (p. 288)**, and **CBMD (p. 287)**.

For complete information, see Using Preintegrated Composite Beam Sections.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Composite Sections

CBTMP, *TEMP*

Specifies a temperature for composite-beam input.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TEMP

Temperature value.

Notes

The **CBTMP (p. 290)** command, one of several composite beam-section commands, specifies a temperature to be associated with the data input via subsequent **CBMX (p. 288)** (preintegrated cross-section stiffness), **CBMD (p. 287)** (preintegrated section mass), or **CBTE (p. 289)** (thermal-expansion) commands.

The specified temperature remains active until the next **CBTMP (p. 290)** command is issued.

An unspecified temperature value defaults to zero.

For complete information, see Using Preintegrated Composite Beam Sections.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Composite Sections

CDOPT, *Option*

Specifies format to be used for archiving geometry.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

IGES

Write solid model geometry information using IGES format (default).

ANF

Write solid model geometry information using Ansys Neutral File (ANF) format.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STAT

Print out the current format setting.

Notes

This command controls your solid model geometry format for **CDWRITE (p. 293)** operations. The ANF option affects only the COMB and SOLID options of the **CDWRITE (p. 293)** command. All other options remain unaffected.

This option setting is saved in the database.

Menu Paths

Main Menu>Preprocessor>Archive Model>Read

Main Menu>Preprocessor>Archive Model>Write

CDREAD,*Option*,*Fname*,*Ext*,--,*Fnamei*,*Exti*

Reads a file of solid model and database information into the database.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Selects which data to read:

ALL

Read all geometry, material property, load, and component data (default). Solid model geometry and loads will be read from the file *fnamei.exti*. All other data will be read from the file *fname.ext*.

DB

Read all database information contained in file *Fname.Ext*. This file should contain all information mentioned above except the solid model loads. If reading a .cdb file written with the GEOM option of the **CDWRITE (p. 293)** command, element types (**ET (p. 686)**) compatible with the connectivity of the elements on the file must be defined prior to reading.

SOLID

Read the solid model geometry and solid model loads from the file *Fnamei.Exti*. This file could have been written by the **CDWRITE (p. 293)** or **IGESOUT (p. 932)** command.

COMB

Read the combined solid model and database information from the file *Fname.Ext*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

The extension defaults to `.cdb` if *Fname* is blank.

--

Unused field.

Fnamei

Name of the IGES file and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Fname*. Used only if *Option* = ALL or SOLID.

Exti

Filename extension (eight-character maximum).

Defaults to IGES if *Fnamei* is blank.

Notes

This command causes coded files of solid model (in IGES format) and database (in command format) information to be read. These files are normally written by the **CDWRITE (p. 293)** or **IGESOUT (p. 932)** command. Note that the active coordinate system in these files has been reset to Cartesian (**CSYS (p. 441)**,0).

If a set of data exists prior to the **CDREAD (p. 291)** operation, that data set is offset upward to allow the new data to fit without overlap. The **NOOFFSET (p. 1250)** command allows this offset to be ignored on a set-by-set basis, causing the existing data set to be overwritten with the new data set.

When you write the geometry data using the **CDWRITE (p. 293)**,GEOM option, you use the **CDREAD (p. 291)**,DB option to read the geometry information.

Using the **CDREAD (p. 291)**,COMB option will not write **NUMOFF (p. 1299)** commands to offset entity ID numbers if there is no solid model in the database.

Multiple `.cdb` file imports cannot have elements with real constants in one file and section definitions in another. The section attributes will override the real constant attributes. If you use **CDREAD (p. 291)** to import multiple CDB files, define all of the elements using only real constants, or using only section definitions. Combining real constants and section definitions is not recommended.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a radiosity mapping data file (`.rsm` file) was saved by the previous **CDWRITE (p. 293)** command, that mapping file must be present in the directory along with the coded geometry file in order for radiosity surface elements (SURF251, SURF252) to be correctly mapped onto the model when **CDREAD (p. 291)** is issued.

If you issue **CDWRITE (p. 293)** to import a `.cdb` file containing a user-defined element, manually insert **USRELEM (p. 2065)** and **USRDOF (p. 2064)** commands in the `.cdb` file to provide the user-defined element characteristics. Insert the two commands *after* the **ET (p. 686)** command (defining the user-defined element) and *before* the EBLOCK command. (If multiple element types are defined in the `.cdb` file, insert the **TYPE (p. 2036)** command to select the user-defined element. Place it *before* **USRELEM (p. 2065)** and **USRDOF (p. 2064)**.)

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Archive Model>Read

CDWRITE, *Option*,*Fname*,*Ext*,--,*Fnamei*, *Exti*,*Fmat*,--,*Key*

Writes geometry and load database items to a file.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Selects which data to write:

ALL

Write all appropriate geometry, material property, load, and component data (default). Two files will be produced. *Fname.Ext* will contain all data items mentioned in "Notes" (p. 295), except the solid model data. *Fnamei.Exti* will contain the solid model geometry and solid model loads data in the form of IGES commands. This option is not valid when **CDOPT (p. 290)**,ANF is active.

COMB

Write all data mentioned, but to a single file, *Fname.Ext*. Solid model geometry data will be written in either IGES or ANF format as specified in the **CDOPT (p. 290)** command, followed by the remainder of the data in the form of Mechanical APDL commands. More information on these (IGES/ANF) file formats is provided in "Notes" (p. 295).

DB

Write all database information except the solid model and solid model loads to *Fname.Ext* in the form of Mechanical APDL commands. This option is not valid when **CDOPT (p. 290)**,ANF is active.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLID

Write only the solid model geometry and solid model load data. This output will be in IGES or ANF format, as specified in the **CDOPT (p. 290)** command. More information on these (IGES/ANF) file formats is provided in "Notes" (p. 295).

GEOM

Write only element and nodal geometry data. Neither solid model geometry nor element attribute data will be written. One file, *Fname.Ext*, will be produced. Use **CDREAD (p. 291)**,DB to read in a file written with this option. Element types (**ET (p. 686)**) compatible with the connectivity of the elements on the file must first be defined before reading the file in with **CDREAD (p. 291)**,DB.

CM

Write only node and element component and geometry data to *Fname.Ext*.

MAT

Write only material property data (both linear and nonlinear) to *Fname.Ext* .

LOAD

Write only loads for current load step to *Fname.Ext*.

SECT

Write only section data to *Fname.Ext*. Pretension sections are not included.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

The extension defaults to CDB if *Fname* is blank.

--

Unused field.

Fnamei

Name of the IGES file and its directory path (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to *Fname*. Used only if *Option* = ALL or SOLID. Previous data on this file, if any, is overwritten.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exti

Filename extension (eight-character maximum).

The extension defaults to IGES in all cases, except when **CDOPT (p. 290)**,ANF is active and **CDWRITE (p. 293)**, *Option* = SOLID. In this case *Exti* = ANF.

Fmat

Format of the output file (defaults to BLOCKED).

BLOCKED

Blocked format. This format allows faster reading of the output file. The time savings is most significant when BLOCKED is used to read `.cdb` files associated with very large models.

UNBLOCKED

Unblocked format.

--

Unused field.

Key

Format key required for Multistage workflow in Workbench.

DUPL

Ensures the correct numbering offsets are applied when writing a `.cdb` file for automatic sector mesh duplication.

Notes

Load data includes the current load step only. Loads applied to the solid model (if any) are automatically transferred to the finite element model when this command is issued. **CDWRITE (p. 293)** writes out solid model loads for meshed models only. If the model is not meshed, the solid model loads cannot be saved. Component data include component definitions, but not assembly definitions. Appropriate **NUMOFF (p. 1299)** commands are included at the beginning of the file; this is to avoid overlap of an existing database when the file is read in.

Solution control commands are typically not written to the file unless you specifically change a default solution setting.

CDWRITE (p. 293) does not support the **GSBDATA (p. 876)** and **GSGDATA (p. 877)** commands, and these commands are not written to the file.

The data may be reread (on a different machine, for example) with the **CDREAD (p. 291)** command. Caution: When the file is read in, the **NUMOFF (p. 1299)**,MAT command may cause a mismatch between material definitions and material numbers referenced by certain loads and element real constants. See **NUMOFF (p. 1299)** for details. Also, be aware that the files created by the **CDWRITE (p. 293)** command explicitly set the active coordinate system to Cartesian (**CSYS (p. 441)**,0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You should generally use the blocked format (*Fmat* = BLOCKED) when writing out model data with **CDWRITE (p. 293)**. This is a compressed data format that greatly reduces the time required to read large models through the **CDREAD (p. 291)** command. The blocked and unblocked formats are described in The CDWRITE (CDB) File Format in the *Programmer's Reference*.

If you use **CDWRITE (p. 293)** in any of the derived products (Ansys Mechanical Pro, Ansys Mechanical Premium), then before reading the file, you must edit the `Jobname.cdb` file to remove commands that are not available in the respective component product.

The **CDWRITE (p. 293)** command does not support (for beam meshing) any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected. For more information about beam meshing, see Meshing Your Solid Model in the *Modeling and Meshing Guide*.

If you issue **CDWRITE (p. 293)** to write a `.cdb` file containing a user-defined element, manually insert **USRELEM (p. 2065)** and **USRDOF (p. 2064)** commands in the `.cdb` file to provide the user-defined element characteristics. Insert the two commands *after* the **ET (p. 686)** command (defining the user-defined element) and *before* the EBLOCK command. (If multiple element types are defined in the `.cdb` file, insert the **TYPE (p. 2036)** command to select the user-defined element. Place it *before* **USRELEM (p. 2065)** and **USRDOF (p. 2064)**.)

If radiosity surface elements (SURF251 or SURF252) are present in the model, a radiosity mapping data file, *Fname*`.RSM`, is also saved when the **CDWRITE (p. 293)** command is issued. For more information, see Advanced Radiosity Options in the *Thermal Analysis Guide*.

IGES and ANF File Formats for Solid Model Geometry Information

The format used for solid model geometry information is determined by the current **CDOPT (p. 290)** command setting. The default format is IGES.

IGES option (default) to write solid model information (**CDOPT (p. 290)**, IGS):

- Before writing solid model entities, select all corresponding lower level entities (**ALLSEL (p. 114)**,BELOW,ALL).
- Section properties assigned to areas, lines and other solid model entities are not maintained when the model is exported.
- If you issue **CDWRITE (p. 293)** after generating a beam mesh with orientation nodes, the database file will contain all of the nodes for every beam element, including the orientation nodes; however, the orientation keypoints that were specified for the line (**LATT (p. 1014)**) are no longer associated with the line and won't be written out to the geometry file. All associativity between the line and the orientation keypoints is lost.
- For beam meshing, this option does not support any line operation that relies on solid model associativity. For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected.
- Concatenated lines are not written. The line segments that make up the concatenated lines are written; however, if the command encounters an area that contains a concatenated line, the write operation halts (that area cannot be recreated during the read operation). If your model has areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that contain concatenated lines, you must first list these and then unconcatenate them before issuing the **CDWRITE (p. 293)** command. Similarly, hardpoint information cannot be written.

ANF option to write solid model information (**CDOPT (p. 290)**, ANF):

- Writes all model information in the database (regardless of select status) to the archive file; however, when you restore the database using this archived file, the select status of entities is also restored.
- Restores all line attributes, including orientation keypoints. It also writes out any components (not assemblies) that consist of solid model entities.
- Halts **CDWRITE (p. 293)** when a concatenated line or an area that contains a concatenated line is detected. You must delete the concatenated lines before issuing **CDWRITE (p. 293)**. Similarly, hardpoint information cannot be written.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Archive Model>Write

CE,*NEQN*,*CONST*,*NODE1*,*Lab1*,*C1*,*NODE2*,*Lab2*,*C2*,*NODE3*,*Lab3*,*C3*

Defines a constraint equation relating degrees of freedom.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NEQN

Set equation reference number:

n

Arbitrary set number.

HIGH

The highest defined constraint equation number. This option is especially useful when adding nodes to an existing set.

NEXT

The highest defined constraint equation number plus one. This option automatically numbers coupled sets so that existing sets are not modified.

The default value is HIGH.

CONST

Constant term of equation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NODE1

Node for first term of equation. If *-NODE1*, this term is deleted from the equation.

Lab1

Degree of freedom label for first term of equation. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Electric labels: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential). Diffusion label: CONC (concentration).

C1

Coefficient for first node term of equation. If zero, this term is ignored.

NODE2, Lab2, C2

Node, label, and coefficient for second term.

NODE3, Lab3, C3

Node, label, and coefficient for third term.

Notes

Repeat the **CE (p. 297)** command to add additional terms to the same equation. To change only the constant term, repeat the command with no node terms specified. Only the constant term can be changed during solution, and only with the **CECMOD (p. 300)** command.

Linear constraint equations may be used to relate the degrees of freedom of selected nodes in a more general manner than described for nodal coupling (**CP (p. 421)**). The constraint equation is of the form:

$$\text{Constant} = \sum_{I=1}^{N} (\text{Coefficient}(I) * U(I))$$

where U(I) is the degree of freedom (displacement, temperature, etc.) of term (I). The following example is a set of two constraint equations, each containing three terms:

0.0 = 3.0* (1 UX) + 3.0* (4 UX) + (-2.0)* (4 ROTY)

2.0 = 6.0* (2 UX) + 10.0* (4 UY) + 1.0* (3 UZ)

The first unique degree of freedom in the equation is eliminated in terms of all other degrees of freedom in the equation. A unique degree of freedom is one which is not specified in any other constraint equation, coupled node set, specified displacement set, or master degree of freedom set. It is recommended that the first term of the equation be the degree of freedom to be eliminated. The first term of the equation cannot contain a master degree of freedom, and no term can contain coupled degrees of freedom. The same degree of freedom may be specified in more than one equation but care must be taken to avoid over-specification (over-constraint).

The degrees of freedom specified in the equation (that is, UX, UY, ROTZ, etc.) must also be included in the model (as determined from the element types (**ET (p. 686)**)). Also, each node in the equation must be defined on an element (any element type containing that degree of freedom will do).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For buckling and modal analyses, the constant term of the equation will not be taken into account (that is, *CONST* is always zero).

Note that under certain circumstances a constraint equation generated by **CE (p. 297)** may be modified during the solution. See Program Modification of Constraint Equations for more information.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Constraint Eqn

CECHECK, *ItemLab*, *Tolerance*, *DOF*

Check constraint equations and couplings for rigid body motions.

PREP7 (p. 22): Database (p. 22)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ItemLab

Item indicating what is to be checked:

CE

Check constraint equations only

CP

Check couplings only

ALL

Check both CE and CP

Tolerance

Allowed amount of out-of-balance for any constraint equation or coupled set. The default value of 1.0e-6 is usually good.

DOF

Specifies which DOF is to be checked. Default is RIGID, the usual option. Other choices are individual DOF such as UX, ROTZ, etc. or THERM. The THERM option will check the constraint equations or coupled sets for free thermal expansions, whereas the individual DOFs check under rigid body motions. ALL is RIGID and THERM.

Notes

This command imposes a rigid body motion on the nodes attached to the constraint equation or coupled set and makes sure that no internal forces are generated for such rigid body motions. Generation of internal forces by rigid body motions usually indicates an error in the equation specification (possibly due to nodal coordinate rotations). The THERM option does a similar check to see that no internal forces

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are created by the equations if the body does a free thermal expansion (this check assumes a single isotropic coefficient of expansion).

Menu Paths

This command cannot be accessed from a menu.

CECMOD, *NEQN*, *CONST*

Modifies the constant term of a constraint equation during solution.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NEQN

Reference number of constraint equation.

CONST

New value of the constant term of equation.

Notes

Other terms of the constraint equation cannot be changed during the solution phase, but must be defined or changed within PREP7 prior to the solution. See the **CE (p. 297)** command for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Modify ConstrEqn

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Modify ConstrEqn

Main Menu>Solution>Load Step Opts>Other>Modify ConstrEqn

CECYCMS, *CycLowNod*, *CycHighNod*, *--*, *KMAP*, *TOLER*, *--*, *KPRINT*, *UsrNMap*

Generates the constraint equations for a multistage cyclic symmetry analysis.

PREP7 (p. 22): Constraint Equations (p. 34)
SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CycLowNod

The name of a component for the nodes located on the low angle edge of the sector (up to 256 characters enclosed in single quotes).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The sector is that of the current stage (*Sname*) specified with **MSOPT (p. 1195)**,NEW,*Sname* or **MSOPT (p. 1195)**,MODIFY,*Sname*. If blank and if the array parameter of edge node pairs does not exist (no user-defined or default for *UsrNMap*), the default component name is '*_Sname*_CYCLOW_NOD'.

CycHighNod

The name of a component for the nodes located on the high angle edge of the sector (up to 256 characters enclosed in single quotes).

The sector is that of the current stage (*Sname*) specified with **MSOPT (p. 1195)**,NEW,*Sname* or **MSOPT (p. 1195)**,MODIFY,*Sname*. If blank and if the array parameter of edge node pairs does not exist (no user-defined or default for *UsrNMap*), the default component name is '*_Sname*_CYCHIGH_NOD'.

--

Unused field.

KMAP

Option to use mapping when creating cyclic symmetry constraint equations. This option is ignored if you specify *UsrNMap*.

ON

Use mapping to relate low and high sector boundary DOFs when applying cyclic symmetry constraint equations.

OFF

Use matching node pairs from low and high sector boundaries to apply cyclic symmetry constraint equations (default).

TOLER

Tolerance for determining if one node on the low edge boundary matches the corresponding node on the high edge boundary after the nodes are rotated.

If positive

TOLER is absolute (length units, defaults to 1e-4). If the distance of the nodes is smaller than this absolute tolerance, the nodes are matched.

If negative

TOLER is relative. Considering the diagonal of an imaginary box enclosing the model, *TOLER* is a fraction of the length of that diagonal. Nodes within the relative tolerance are matched.

--

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KPRINT

Option to print the table of matched nodes (*KMAP* = OFF) or mapped nodes and elements (*KMAP* = ON).

0

Do not print the table (default).

1

Print the table. If edge nodes are mapped (*KMAP* = ON) and a high edge node is matching a low edge node, the third column labeled "MAPPED" lists the node number. (See Snippets of Table Printed with *KPRINT* = 1 on **CECYCMS (p. 300)** (p. 303)).

UsrNMap

Option for matching node pairs between low and high edges.

Input the name of an existing array parameter or a numerical key:

<name>

Name of a user-defined array parameter that specifies the matching node pairs. The node pairs in the parameter may be input in any order, but the low edge node must be the first entry in each pair. (See Example: **CECYCMS (p. 300)** with a User-defined Array Parameter for *UsrNMap* (p. 303).)

0 (*or* blank)

If the default array parameter named *_Sname*_CYCNODPAIR already exists, it is used to specify the matching node pairs (default).

If this array parameter does not exist, nodes are paired automatically, and the array parameter named *_Sname*_CYCNODPAIR is created.

1

Nodes are paired automatically, and the array parameter named *_Sname*_CYCNODPAIR is created. If it exists, it is deleted and re-created.

-1

Nodes are paired automatically without creating or using an array parameter.

Notes

CECYCMS (p. 300), **CEIMS (p. 306)**, and **MSOPT (p. 1195)** are commands used in a multistage cyclic symmetry analysis.

If edge node pairs are matched (*KMAP* = OFF) and an array parameter is not specified for *UsrNMap*, components are used for the cyclic edge nodes. You must specify those components using the **CM (p. 356)** command and ensure that they contain base sector nodes only. See Building the Model for a more

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

detailed discussion and Example Usage (p. 303) for examples demonstrating the use of **CECYCMS (p. 300)** in multistage cyclic symmetry analyses.

Snippets of Table Printed with *KPRINT* = 1 on CECYCMS (p. 300)

- If *KMAP* = OFF, matched nodes are printed:

```
*****  CECYCMS - NODE PAIRS  *****
          nLow =       305 nHigh =       293
          nLow =       477 nHigh =       471
          nLow =       478 nHigh =       472
          nLow =       479 nHigh =       473
          nLow =       480 nHigh =       474
          nLow =       481 nHigh =       475
          nLow =       482 nHigh =       476
          nLow =       279 nHigh =       270
          nLow =       601 nHigh =       615
```

- If *KMAP* = ON, matched mapped nodes and elements are printed:

```
********* MAPPED NODES AND ELEMENTS *********
            NODE    ELEMENT    MATCHED NODE
            262        481          271
            263        487          272
            264        493          273
            265        499          274
            266        505          275
            267        511          276
            268        517          277
            269        523          278
```

Example: CECYCMS (p. 300) with a User-defined Array Parameter for *UsrNmap*

```
*DIM,MYMAP,ARRAY,2,10          ! specifying 10 low-high edge node pairs
*set,mymap(1, 1), 107, 108     ! low node 107 <> high node 108
*set,mymap(1, 2), 147, 211     ! low node 147 <> high node 211
…
CECYCMS,,, ,,, ,,MYMAP
```

Example Usage

Example: Static Analysis of a Compressor Model with 4 Axial Stages Without a Duplicate Sector

Example: Linear Perturbation Modal Analysis of a Simplified Model with 2 Axial Stages and a Non-planar Interstage Boundary

Example: Modal Analysis of Turbomachinery Stage Modeled as 2 Radial Stages with Offset Cyclic Edge Starting Points

Example: Mutistage Multiharmonic Modal Analysis of a Hollow Cylinder Modeled Using 2 Stages

Example: Multiharmonic Linear Perturbation Modal Analysis of a Simplified Model with 3 Axial Stages

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CEDELE, *NEQN1*, *NEQN2*, *NINC*, *Nsel*

Deletes constraint equations.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NEQN1*, *NEQN2*, *NINC

Delete constraint equations from *NEQN1* to *NEQN2* (defaults to *NEQN1*) in steps of *NINC* (defaults to 1). If *NEQN1* = ALL, *NEQN2* and *NINC* will be ignored all constraint equations will be deleted.

Nsel

Additional node selection control:

ANY

Delete equation set if *any* of the selected nodes are in the set (default).

ALL

Delete equation set only if *all* of the selected nodes are in the set.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Del Constr Eqn

CEFILTER, *TermKey*

Controls the automatic solution termination of models having constraint equations with too many terms.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TermKey

Enables or disables the solution termination if a model has constraint equations with too many terms:

ON

Automatically terminates the simulation with an error message (see Notes) upon encountering a model that has constraint equations with too many terms (default).

OFF

Disables automatic termination, and instead allows the solution to continue even if a model has constraint equations with too many terms.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

A model with one or more constraint equations with too many terms is ill-posed, requires excessive memory allocation and solution times, and may never converge. By default, the program automatically terminates the simulation of such models with an error message stating that there were too many constraint equation terms. One reason this can occur is the misuse of Remote Points in the Mechanical Application, which create MPC equations. Best practice is to revise your model to have constraint equations with less terms.

If, instead, you wish to allow the solution to continue with increased solution time and memory requirements without changing your model, issue **CEFILTER (p. 304)**,OFF to bypass the automatic termination.

Menu Paths

This command cannot be accessed from a menu.

CEINTF, *TOLER*, *DOF1*, *DOF2*, *DOF3*, *DOF4*, *DOF5*, *DOF6*, *MoveTol*

Generates constraint equations at an interface.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TOLER

Tolerance about selected elements, based on a fraction of the element dimension (defaults to 0.25 (25%)). Nodes outside the element by more than the tolerance are not accepted as being on the interface.

DOF1*, *DOF2*, *DOF3*, . . . , *DOF6

Degrees of freedom for which constraint equations are written. Defaults to all applicable DOFs. *DOF1* accepts ALL as a valid label, in which case the rest are ignored (all DOFs are applied).

MoveTol

The allowed "motion" of a node (see Note below). This distance is in terms of the element coordinates (-1.0 to 1.0). A typical value is 0.05. Defaults to 0 (do not move). *MoveTol* must be less than or equal to *TOLER*.

Notes

This command can be used to "tie" together two regions with dissimilar mesh patterns by generating constraint equations that connect the selected nodes of one region to the selected elements of the other region. At the interface between regions, nodes should be selected from the more dense mesh region, A, and the elements selected from the less dense mesh region, B. The degrees of freedom of region A nodes are interpolated with the corresponding degrees of freedom of the nodes on the region B elements, using the shape functions of the region B elements. Constraint equations are then written that relate region A and B nodes at the interface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The *MoveTol* field lets the nodes in the previously mentioned region A change coordinates when slightly inside or outside the elements of region B. The change in coordinates causes the nodes of region A to assume the same surface as the nodes associated with the elements of region B. The constraint equations that relate the nodes at both regions of the interface are then written.

Solid elements with six degrees of freedom should only be interfaced with other six degree-of-freedom elements. The region A nodes should be near the region B elements. A location tolerance based on the smallest region B element length may be input. Stresses across the interface are not necessarily continuous. Nodes in the interface region should not have specified constraints.

Use the **CPINTF (p. 425)** command to connect nodes by coupling instead of constraint equations. Use the **EINTF (p. 614)** command to connect nodes by line elements. See also the **NSEL (p. 1266)** and **ESEL (p. 661)** commands for selecting nodes and elements. See the *Mechanical APDL Theory Reference* for a description of 3D space used to determine if a node will be considered by this command.

As an alternative to the **CEINTF (p. 305)** command, you can use contact elements and the internal multipoint constraint (MPC) algorithm to tie together two regions having dissimilar meshes. See Solid-Solid and Shell-Shell Assemblies for more information.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Adjacent Regions

CEIMS,*TOLER*,--,*Sname1*,*Sname2*,*KPRINT*,--,--,--,--,*IntF1Nod*,*IntF2Nod*

Generates constraint equations at the interstage boundary in a multistage cyclic symmetry analysis.

PREP7 (p. 22): Constraint Equations (p. 34)
SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Note:

CEIMS (p. 306) command is not part of the **CYCLIC (p. 462)** procedure for cyclic symmetry analysis. This command is used in a multistage cyclic symmetry analysis (see also **MSOPT (p. 1195)**) to generate interstage constraint equations.

TOLER

Tolerance for determining if selected nodes are on the interface. *TOLER* is a fraction of the element dimension (defaults to 0.25 (25%)). Nodes outside the element by more than the tolerance are not accepted as being on the interface.

--

Unused field.

Sname1

The name of the first stage or part. For details on required node and element selections, see Node and Element Selection Requirements (p. 307).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sname2

The name of the second stage or part. For details on required node and element selections, see Node and Element Selection Requirements (p. 307).

KPRINT

Option to print mapped nodes and elements.

0 (*or* OFF)

Do not print mapped nodes and elements (default).

1 (*or* ON)

Print mapped nodes and elements.

--, --, --, --

Unused fields.

IntF1Nod

The name of the interstage nodal component of the first stage or sector part to be tied to the second stage or part named *IntF2Nod*. It is optional to specify *IntF1Nod* (see Node and Element Selection Requirements (p. 307)), but if used, *IntF2Nod* must also be specified.

IntF2Nod

The name of the interstage nodal component of the second stage or sector part to be tied to the first stage or part named *IntF1Nod*. It is optional to specify *IntF2Nod* (see Node and Element Selection Requirements (p. 307)), but it used, *IntF1Nod* must also be specified.

Notes

This command can be used to generate constraint equations to tie the interface nodes of two cyclic sector parts.

Mapping is performed so mesh patterns at the interface of both parts can be different.

This command is supported for the following degrees of freedom (DOFs) at the interstage boundary: UX, UY, UZ, ROTX, ROTY, ROTZ. Since only 3D elements are supported, UX, UY, and UZ are required. Note that if rotational DOFs are included, all three of them must be present.

Node and Element Selection Requirements

You can specify the interface nodes in one of two ways:

1. If you specify *IntF1Nod* and *IntF2Nod* nodal components, there are no other node or element selection requirements.

2. Otherwise, prior to issuing CEIMS:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- the **nodes** at the interface of the first cyclic sector part (*Sname1*, part having the largest cyclic sector angle) must be selected*, and*
- the **elements** at the interface of the second cyclic sector part (*Sname2*, part having the smallest cyclic sector angle) must be selected.

For cyclic sector parts, select only base sector quantities (not duplicate sector ones). See also the **NSEL (p. 1266)** and **ESEL (p. 661)** commands for selecting nodes and elements.

The degrees of freedom of the first part interface nodes are interpolated with the corresponding degrees of freedom of the nodes of the second part interface elements using the shape functions of those elements.

Constraint equations are created between interface nodes. Those nodes should not have any other constraints defined, but if so they must be compatible.

Example Usage

Example: Static Analysis of a Compressor Model with 4 Axial Stages Without a Duplicate Sector

Example: Linear Perturbation Modal Analysis of a Simplified Model with 2 Axial Stages and a Non-planar Interstage Boundary

Example: Modal Analysis of Turbomachinery Stage Modeled as 2 Radial Stages with Offset Cyclic Edge Starting Points

Example: Mutistage Multiharmonic Modal Analysis of a Hollow Cylinder Modeled Using 2 Stages

Example: Multiharmonic Linear Perturbation Modal Analysis of a Simplified Model with 3 Axial Stages

Menu Paths

This command cannot be accessed from a menu.

CELIST,*NEQN1*,*NEQN2*,*NINC*,*Option*

Lists the constraint equations.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NEQN1*, *NEQN2*, *NINC

List constraint equations from *NEQN1* to *NEQN2* (defaults to *NEQN1*) in steps of *NINC* (defaults to 1). If *NEQN1* = ALL (default), *NEQN2* and *NINC* are ignored and all constraint equations are listed.

Option

Options for listing constraint equations:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANY

List equation set if *any* of the selected nodes are in the set (default). Only externally-generated constraint equations are listed.

ALL

List equation set only if *all* of the selected nodes are in the set. Only externally-generated constraint equations are listed.

INTE

List internally-generated constraint equations that are associated with MPC-based contact. Constraint equations are listed only if *all* the nodes in the set are selected.

CONV

Convert internal constraint equations to external constraint equations. Internal constraint equations are converted only if *all* of the nodes in the set are selected.

Notes

This command is valid in any processor. However, the INTE and CONV options are only valid in the Solution processor after a **SOLVE (p. 1822)** command has been issued.

Menu Paths

Utility Menu>List>Other>Constraint Eqns>All CE nodes selected

Utility Menu>List>Other>Constraint Eqns>Any CE node selected

CENTER, *NODE*, *NODE1*, *NODE2*, *NODE3*, *RADIUS*

Defines a node at the center of curvature of 2 or 3 nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Number to be assigned to the node generated at the center of curvature.

NODE1, NODE2, NODE3

Three nodes used to calculated the center of curvature, as described under *RADIUS*.

RADIUS

Used to control the interpretation of *NODE1*, *NODE2* and *NODE3*:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

NODE1, *NODE2* and *NODE3* lie on a circular arc. The program will calculate the center of curvature (and radius) (default).

≠0

NODE1 and *NODE2* are the endpoints of an arc, and *RADIUS* is the radius of curvature. The program will locate the center of curvature on the *NODE3* side of the *NODE1-NODE2* line if *RADIUS* > 0, and opposite to *NODE3* if *RADIUS* < 0.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>At Curvature Ctr

---

CEQN

Specifies "Constraint equations" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Constraint Eqns

CERIG, *INDEPEND*, *DEPEND*, *Ldof*, *Ldof2*, *Ldof3*, *Ldof4*, *Ldof5*

Defines a rigid region.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

INDEPEND

Retained (or independent) node for this rigid region. If *INDEPEND* = P, then graphical picking of the independent and dependent nodes is enabled (first node picked will be the independent node, and subsequent nodes picked will be dependent nodes), and subsequent fields are ignored (valid only in GUI).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEPEND

Removed (or dependent) node for this rigid region. If ALL, dependent nodes are all selected nodes.

Ldof

Degrees of freedom associated with equations:

ALL

All applicable degrees of freedom (default). If 3D, generate 6 equations based on UX, UY, UZ, ROTX, ROTY, ROTZ; if 2D, generate 3 equations based on UX, UY, ROTZ.

UXYZ

Translational degrees of freedom. If 3D, generate 3 equations based on the dependent nodes' UX, UY, and UZ DOFs and the independent node's UX, UY, UZ, ROTX, ROTY, and ROTZ DOFs. If 2D, generate 2 equations based on the dependent nodes UX and UY DOFs and the independent nodes UX, UY, and ROTZ DOFs. No equations are generated for the rotational coupling.

RXYZ

Rotational degrees of freedom. If 3D, generate 3 equations based on ROTX, ROTY, ROTZ; if 2D, generate 1 equation based on ROTZ. No equations are generated for the translational coupling.

UX

Dependent translational UX degree of freedom only.

UY

Dependent translational UY degree of freedom only.

UZ

Dependent translational UZ degree of freedom only.

ROTX

Dependent rotational ROTX degree of freedom only.

ROTY

Dependent rotational ROTY degree of freedom only.

ROTZ

Dependent rotational ROTZ degree of freedom only.

Ldof2*, *Ldof3*, *Ldof4*, *Ldof5

Additional degrees of freedom. Used only if more than one degree of freedom are required and *Ldof* is not ALL, UXYZ, or RXYZ.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines a rigid region (link, area or volume) by automatically generating constraint equations to relate nodes in the region. Nodes in the rigid region must be assigned a geometric location before this command is used. Also, nodes must be connected to elements having the required degree of freedom set (see *Ldof* above). Generated constraint equations are based on small deflection theory. Generated constraint equations are numbered beginning from the highest previously defined equation number (*NEQN*) plus 1. Equations, once generated, may be listed (**CELIST (p. 308)**) or modified (**CE (p. 297)**) as desired. Repeat the **CERIG (p. 310)** command for additional rigid region equations.

This command generates the constraint equations needed for defining rigid lines in 2D or 3D space. Multiple rigid lines relative to a common point are used to define a rigid area or a rigid volume. In 2D space, with *Ldof* = ALL, three equations are generated for each pair of constrained nodes. These equations define the three rigid body motions in global Cartesian space, that is, two in-plane translations and one in-plane rotation. These equations assume the X-Y plane to be the active plane with UX, UY, and ROTZ degrees of freedom available at each node. Other types of equations can be generated with the appropriate *Ldof* labels.

Six equations are generated for each pair of constrained nodes in 3D space (with *Ldof* = ALL). These equations define the six rigid body motions in global Cartesian space. These equations assume that UX, UY, UZ, ROTX, ROTY, and ROTZ degrees of freedom are available at each node.

The UXYZ label allows generating a partial set of rigid region equations. This option is useful for transmitting the bending moment between elements having different degrees of freedom at a node. With this option only two of the three equations are generated for each pair of constrained nodes in 2D space. In 3D space, only three of the six equations are generated. In each case the rotational coupling equations are not generated. Similarly, the RXYZ label allows generating a partial set of equations with the translational coupling equations omitted.

Applying this command to a large number of dependent nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of dependent nodes.

Note that under certain circumstances the constraint equations generated by **CERIG (p. 310)** may be modified during the solution. See Program Modification of Constraint Equations for more information.

CERIG (p. 310) is restricted to small-deflection analysis (large-deflection is not supported). As an alternative to the **CERIG (p. 310)** command, you can define a similar type of rigid region using contact elements and the internal multipoint constraint (MPC) algorithm. See Surface-Based Constraints for more information.

CERIG (p. 310) cannot be deleted using **CEDELE (p. 304)**,ALL and then regenerated in the second or higher load steps if the **LSWRITE (p. 1087)** and **LSSOLVE (p. 1085)** procedure is used. **CERIG (p. 310)** writes constraint equations directly into load step files. Deleting constraint equations (**CEDELE (p. 304)**,ALL) cannot always maintain the consistency among load steps.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Rigid Region

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CESEL, *Type*, - - , - - , *VMIN*, *VMAX*, *VINC*

Selects constraint equations via predefined reference numbers.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set (default).

A

Select an additional set and add it to the current set.

U

Unselect a set from the current set.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

- - , - -

Unused fields.

The following fields are used only with *Type* = S, A, or U:

VMIN

Minimum value of constraint equation reference number range.

VMAX

Maximum value of constraint equation reference number range. *VMAX* defaults to *VMIN*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VINC

Value increment within the specified range. Defaults to 1.

Command Default

All constraint equations are selected.

Notes

The **CESEL (p. 313)** command selects sets of constraint equations (**CE (p. 297)**) via specified reference numbers. *VMIN*, *VMAX*, and *VINC* must be positive integer values.

For example, the following command selects a new set of constraint equations based on reference numbers 1 through 7:

```
CESEL,S,,,1,7,1
```

Data are flagged as selected and unselected; no data are actually deleted from the database.

Use **CELIST (p. 308)** to list constraint equations and their reference numbers. If a constraint equation is selected but involves unselected nodes, that constraint equation will not be listed by the **CELIST (p. 308)** command, and the solver ignores it.

Internal constraint equations are not affected by this command.

This command is also valid in POST1.

Menu Paths

This command cannot be accessed from a menu.

CESGEN, *ITIME*, *INC*, *NSET1*, *NSET2*, *NINC*

Generates a set of constraint equations from existing sets.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME*, *INC

Do this generation operation a total of *ITIME*s, incrementing all nodes in the existing sets by *INC* each time after the first. *ITIME* must be >1 for generation to occur.

NSET1*, *NSET2*, *NINC

Generate sets from sets beginning with *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* is negative, *NSET2* and *NINC* are ignored and the last |*NSET1*| sets (in sequence from maximum set number) are used as the sets to be repeated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates additional sets of constraint equations (with same labels) from existing sets. Node numbers between sets may be uniformly incremented.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same DOF

CFACT, *RFACTA*, *IFACTA*, *RFACTB*, *IFACTB*, *RFACTC*, *IFACTC*

Defines complex scaling factors to be used with operations.

POST26 (p. 58): Controls (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RFACTA

Real portion of the complex scale factor used in place of *FACTA*.

IFACTA

Imaginary portion of the complex scale factor used in place of *FACTA*.

RFACTB

Real portion of the complex scale factor used in place of *FACTB*.

IFACTB

Imaginary portion of the complex scale factor used in place of *FACTB*.

RFACTC

Real portion of the complex scale factor used in place of *FACTC*.

IFACTC

Imaginary portion of the complex scale factor used in place of *FACTC*.

Command Default

Use the real factors as described with the operation command.

Notes

Defines complex scale factors to be used with the operations (**ADD (p. 95)**, **PROD (p. 1515)**, etc.). If this command is supplied, these complex factors override any real factors (*FACTA*, *FACTB*, *FACTC*) supplied on the operation commands. Factors are typically involved in scaling a specified variable, such as in the term *FACTA* x *IA* of the **ADD (p. 95)** command to scale variable *IA* before the ADD operation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **CFACT (p. 315)** command is active, defaults are as follows: 1) if the complex factor is not specified, but the variable upon which it acts (such as *IA*) is specified, the factor defaults to 1.0+*i*0.0; 2) if the variable upon which the factor operates is not specified, but the factor is specified, the variable defaults to 1.0 so that the term in the operation becomes the complex factor itself; 3) if neither the factor nor the variable number is supplied, the term is omitted from the operation. Once the operation (such as the **ADD (p. 95)** command) has been processed, the **CFACT (p. 315)** command becomes inactive and must be specified again if it is to be used.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Complx ScaleFact

---

*CFCLOS

Closes the "command" file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

---

*CFOPEN, *Fname*, *Ext*, --, *Loc*

Opens a "command" file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to CMD if *Fname* is blank.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--

Unused field.

Loc

Determines whether existing file will be overwritten or appended:

(blank) --

The existing file will be overwritten.

APPEND --

The file will be appended to the existing file.

Notes

Mechanical APDL commands specified by the ***CFWRITE (p. 318)** command are written to the file opened by ***CFOPEN (p. 316)**. Data processed with the ***VWRITE (p. 2174)** command are also written to this file if the file is open when the ***VWRITE (p. 2174)** command is issued.

Issue the ***CFCLOS (p. 316)** command to close the command file.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

/CFORMAT, *NFIRST*, *NLAST*

Controls the graphical display of alphanumeric character strings for parameters, components, assemblies, and tables.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFIRST

Display the first *n* characters of the parameter, component, assembly, or table name, up to 32. Defaults to 32.

NLAST

Display the last *n* characters of the parameter, component, assembly, or table name, up to 32. Defaults to 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Use this command to control the length of the character string that is shown in the graphics window for a parameter, component, assembly, or table name.

The total number of characters (*NFIRST* + *NLAST* +3) cannot exceed 32.

If *NFIRST* is greater than zero and *NLAST* = 0, only the *NFIRST* characters are displayed, followed by an ellipsis.

If *NFIRST* = 0 and *NLAST* is greater than zero, only the *NLAST* characters are displayed, preceded by an ellipsis (...).

If both *NFIRST* and *NLAST* are greater than zero, the name will be shown as *NFIRST*, followed by an ellipsis (...), followed by *NLAST*, up to a maximum of 32 characters.

For example, if *NFIRST* = 6 and *NLAST* = 3, and the character string is LENGTHOFSIDEONE, then it will appear in the graphics window as LENGTH...ONE.

If the actual length of the character string is less than the specified combination of *NFIRST* + *NLAST* +3, then the actual string will be used.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

***CFWRITE**, *Command*

Writes a Mechanical APDL command (or similar string) to a "command" file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Command

Command or string to be written. The standard command form of a label followed by arguments separated by commas is assumed. *Command* may be a parameter assignment (for example, ***CFWRITE (p. 318)**, A = 5).

Notes

Writes a Mechanical APDL command (or similar string) to the file opened via ***CFOPEN (p. 316)**. The *Command* string is not executed (except that numeric and character parameter substitution and operations (with imbedded *, /, >, etc. characters) are performed before writing).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When used with ***GET (p. 797)** results and parameter substitution, a command can be created from results and then read back into the Mechanical APDL program (or used elsewhere). For example, if the command ***CFWRITE (p. 318)**,BF,NNUM,TEMP,TVAL is used in a do-loop, where TVAL is a parameter value returned from the ***GET (p. 797)** operation and NNUM is a specified or returned parameter value, a series of **BF (p. 221)** commands, with numerical values substituted for the two parameters, will be written.

To create a file without parameter substitution, issue ***CREATE (p. 434)**.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

CGLOC, *XLOC*, *YLOC*, *ZLOC*

Specifies the origin location of the acceleration coordinate system.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XLOC*, *YLOC*, *ZLOC

Global Cartesian X, Y, and Z coordinates of the acceleration coordinate system origin.

Notes

Specifies the origin location of the acceleration coordinate system with respect to the global Cartesian system. The axes of this acceleration coordinate system are parallel to the global Cartesian axes.

A structure may be rotating about the global Cartesian origin (**OMEGA (p. 1324)**, **DOMEGA (p. 571)**), which may in turn be rotating about another point (the origin of the acceleration coordinate system), introducing Coriolis effects. The location of this point (relative to the global Cartesian origin) is specified with this **CGLOC (p. 319)** command. For example, if Y is vertical and the global system origin is at the surface of the earth while the acceleration system origin is at the center of the earth, *YLOC* should be -4000 miles (or equivalent) if the rotational effects of the earth are to be included. The rotational velocity of the global Cartesian system about this point is specified with the **CGOMGA (p. 320)** command, and the rotational acceleration is specified with the **DCGOMG (p. 501)** command.

The rotational velocities and accelerations are mainly intended to include mass effects in a static (**ANTYPE (p. 162)**,STATIC) analysis. If used in dynamic analyses, no coupling exists between the user input terms and the time history response of the structure. See Acceleration Effect in the *Mechanical APDL Theory Reference* for details. Related commands are **ACEL (p. 92)**, **CGOMGA (p. 320)**, **DCGOMG (p. 501)**, **DOMEGA (p. 571)**, and **OMEGA (p. 1324)**.

See Analysis Tools in the *Theory Reference* for more information.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects

Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects

---

CGOMGA, *CGOMX*, *CGOMY*, *CGOMZ*

Specifies the rotational velocity of the global origin.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CGOMX*, *CGOMY*, *CGOMZ

Rotational velocity of the global origin about the acceleration system X, Y, and Z axes.

Notes

Specifies the rotational velocity of the global origin about each of the acceleration coordinate system axes. The location of the acceleration coordinate system is defined with the **CGLOC (p. 319)** command. Rotational velocities may be defined in these analysis types:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC) -- full, VT [1], Krylov [1], or mode-superposition [2]
- Transient (**ANTYPE (p. 162)**,TRANS) -- full or mode-superposition [2]
- Substructuring (**ANTYPE (p. 162)**,SUBSTR)
- Modal (**ANTYPE (p. 162)**,MODAL)

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

See Acceleration Effect in the *Mechanical APDL Theory Reference* for details. Units are radians/time. Related commands are **ACEL (p. 92)**, **CGLOC (p. 319)**, **DCGOMG (p. 501)**, **DOMEGA (p. 571)**, and **OMEGA (p. 1324)**.

See Analysis Tools in the *Theory Reference* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **CGOMGA (p. 320)** command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *CGOMGA_X*, *CGOMGA_Y*, and *CGOMGA_Z* input values (***DIM (p. 530)**) for full transient and harmonic analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects

Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects

---

CGROW,*Action*,*Par1*,*Par2*,*Par3*,*Par4*

Specifies crack-growth options in a fracture analysis.

PREP7 (p. 22): Data Tables (p. 24)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Specifies the action for defining or manipulating crack-growth data:

NEW (p. 322)

Initiate a new set of crack-growth simulation data (default).

CID (p. 323)

Specify the crack-calculation (**CINT (p. 334)**) ID for energy-release rates to be used in the fracture criterion calculation.

CGID (p. 323)

Activate an existing crack-growth ID to modify its crack-growth settings via subsequent **CGROW (p. 321)** commands.

FCOPTION (p. 323)

Specify the fracture criterion for crack-growth/delamination.

CSFL (p. 324)

Specify a crack-surface load for crack-growth.

CPATH (p. 325)

Specify the element component for crack-growth.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DTIME (p. 325)

Specify the initial time step for crack-growth.

DTMIN (p. 325)

Specify the minimum time step for crack-growth.

DTMAX (p. 325)

Specify the maximum time step for crack-growth.

FCRAT (p. 325)

Fracture criterion ratio (fc).

STOP (p. 326)

Stops the analysis when the specified criterion (*Par1*) is reached.

METHOD (p. 326)

Define the method of crack propagation.

FCG (p. 327)

Specifies fatigue crack-growth.

SOPT (p. 328)

Specifies crack-growth solution options.

RMCONT (p. 330)

Specifies remeshing control options for mesh coarsening.

NPLOAD (p. 331)

Specifies nonproportional loading control options for fatigue crack-growth.

Command Specification for *Action* = NEW

CGROW (p. 321), NEW, *Par1*

Par1 –

Crack-growth data set ID (integer value).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Action* = CID

CGROW (p. 321), CID, *Par1*

Par1 –

Contour-integral calculation (**CINT (p. 334)**) ID for energy-release rates to be used in fracture criterion calculation.

Command Specification for *Action* = CGID

CGROW (p. 321), CGID, *Par1*

Par1 –

Crack-growth data set ID (integer value).

Command Specification for *Action* = FCOPTION

CGROW (p. 321), FCOPTION, *Par1*, *Par2*, *Par3*

Par1 –

MTAB – Crack-growth fracture criterion used with the material data table (**TB (p. 1899)**,CGCR (p. 1912)).

GTC – Defines the critical energy-release rate, a simple fracture criterion used for the VCCT method.

KIC – Defines the critical stress-intensity factor. Valid for the SMART method only.

JIC – Defines the critical J-integral. Valid for the SMART method only.

Par2 –

For *Par1* = MTAB: Material ID for the material data table.

For I*Par1* = GTC: Critical energy-release rate value.

For *Par1* = KIC: Critical stress-intensity factor value.

For *Par1* = JIC: Critical J-integral value.

Par3 –

For *Par1* = MTAB, KIC, or JIC: Specifies the fracture-parameter contour to use for SMART crack-growth evaluation. Default = 2.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Action* = CSFL

CGROW (p. 321), CSFL, *Par1*, *Par2*, *Par3*

Par1 –

Crack-surface load options:

PRES – New crack-surface pressure load.

CZM – Entire crack-surface enhanced by INTER204 3D 16-node cohesive elements (generated automatically during the first remeshing around the associated crack). See "Notes" (p. 332).

Par2 –

When *Par1* = PRES:

When *Par1* = CZM: A valid material ID (defined via **TB (p. 1899)**,CZM,,,,BILI, **TB (p. 1899)**,CZM,,,,CEXP, or **TB (p. 1899)**,CZM,,,,CLIN [default]). If zero or an invalid value is specified, the program uses **TB (p. 1899)**,CZM,,,,CLIN as the default cohesive model type and sets reasonable CZM parameters based on the solid elements around the crack fronts/surfaces. For more information, see Enhancing Crack Surfaces with Cohesive Zone Elements in the *Fracture Analysis Guide*.

Par3 –

When *Par1* = PRES:

New crack-surface load value, or the name of a table for specifying tabular load values for crack-growth.

The new crack-surface load is always assumed to be constant. If *Par1* = PRES, *Par3* specifies a constant pressure load.

To specify a table (***DIM (p. 530)**), enclose the table name within "%" characters (**%*tablename*%**). Only one table can be specified for a crack-growth set, and time is the only primary variable supported.

When *Par1* = CZM:

A valid local coordinate system (LCS) ID (**LOCAL (p. 1059)**).

If zero or an invalid value is specified, the program creates an internal LCS and assigns it to the ESYS attributes of CZM elements inserted on crack surfaces. This behavior applies only when the material ID (*MAT_ID*) for *Par3* is associated with **TB (p. 1899)**,CZM when *TBOPT* = FRIC or REXP. For more information, see Enhancing Crack Surfaces with Cohesive Zone Elements in the *Fracture Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Action* = CPATH

CGROW (p. 321), CPATH, *Par1*

Par1 –

Interface element component for crack path (VCCT method only). (Component name must be 32 characters or less.)

Command Specification for *Action* = DTIME

CGROW (p. 321), DTIME, *Par1*

Par1 –

Initial time step when crack-growth is detected.

Command Specification for *Action* = DTMIN

CGROW (p. 321), DTMIN, *Par1*

Par1 –

Minimum time step allowed when crack-growth is detected.

Command Specification for *Action* = DTMAX

CGROW (p. 321), DTMAX, *Par1*

Par1 –

Maximum time step allowed when crack-growth is detected.

Command Specification for *Action* = FCRAT

CGROW (p. 321), FCRAT, *Par1*

Par1 –

Fracture criterion ratio (f_c, where f_c is generally around 1). The recommended ratio is 0.95 to 1.05. The default is 1.00.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Action* = STOP

CGROW (p. 321), STOP, *Par1*, *Par2*

***Par1* –**

CEMX – Stops the analysis when the crack extension for any crack-front node reaches the maximum value (*Par2*).

CNMX – Stops the analysis when the maximum number of fatigue cycles exceeds the specified limit (*Par2*). Valid only for SMART crack-growth analysis.

FBOU – Stops the analysis when the crack extension reaches the free boundary. Valid only for SMART crack-growth analysis.

KMAX – Stops the analysis when the maximum equivalent stress-intensity factor exceeds the specified limit (*Par2*). Valid only for SMART crack-growth analysis.

DN – Stops the analysis when the fatigue-cycle increment number is less than or equal to the specified limit (*Par2*). Valid only for SMART crack-growth analysis.

***Par2* –**

When *Par1* = CEMX, the value of maximum crack extension allowed.

When *Par1* = CNMX, the maximum number of fatigue cycles allowed.

When *Par1* = KMAX, the value of the maximum equivalent stress-intensity factor allowed.

When *Par1* = DN, the minimum limit for fatigue-cycle increment. Default = 0.

Command Specification for *Action* = METHOD

CGROW (p. 321), METHOD, *Par1*

***Par1*, *Par2* –**

VCCT – Use VCCT to grow the crack (default).

XFEM – Use XFEM to grow the crack.

SMART– Use SMART to grow the crack.

***Par2* –**

REME – Remeshing-based SMART crack-growth method (the default and only option). Valid only when *Par1* = SMART.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Action* = FCG

CGROW (p. 321), FCG, *Par1*,*Par2*,*Par3*

***Par1* –**

METH – Fatigue crack-growth method.

DAMX – Maximum crack-growth increment (XFEM method only).

DAMN – Minimum crack-growth increment (XFEM method only).

SRAT – Stress ratio.

DKTH – Threshold value of equivalent stress-intensity-factor range (SIFS).

DN – Incremental number of cycles.

***Par2* –**

Method – LC or CBC method. Valid only when *Par1* = METH.

VALUE – Value for the specified *Par1*. Valid only when *Par1* = DAMX, DAMN, SRAT, DKTH, or DN.

For *Par1* = SRAT or DKTH, *Par2* can be a constant or table. To specify a table (***DIM (p. 530)**), enclose the table name within "%" characters (%*tablename*%). Only one table can be specified for a crack-growth set, and time is the only primary variable supported.

***Par3* –**

Valid only for SMART crack-growth analysis when *Par1* = DKTH.

MIN *or* MAX *or* AVER *or* blank (default) – The minimum, maximum, or average stress-intensity-factor range (ΔK), respectively. The program calculates the range over the entire crack-front, then uses the result to evaluate the threshold criterion. The entire crack is arrested if the specified minimum, maximum, or average ΔK is below the threshold (*Par2*). Partial crack-growth does not occur.

If *Par3* is unspecified (default), the program evaluates threshold criterion at each crack-front node individually and ensures that the crack is arrested locally at any node where ΔK is below the threshold (*Par2*). If some nodes have ΔK less than the threshold and others have ΔK greater than the threshold, partial crack-growth occurs at the crack-front.

Example 2: Using CGROW, FCG, DKTH, *Par2*, *Par3*

If *Par3* = MIN, the program selects the minimum ΔK from the ΔK values calculated at all crack-front nodes for the corresponding crack ID. If the minimum ΔK does not exceed the threshold (*Par2*), there is no crack extension at the entire crack-front.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Action* = SOPT

CGROW (p. 321),SOPT,*Par1*,*Par2*,*Par3*,*Par4*

Par1 **–**

SCN – Use a single number of cycles for all cracks in a multicrack analysis (even if each crack has separate loading) (default).

MCN – Allow a separate cycle count for *each* crack in a multicrack analysis (available only if each crack has separate loading).

MXAN – Specifies the maximum limit of the crack-growth kinking angle.

MNAN – Specifies the cut-off value for the minimum crack-growth kinking angle.

DAMX – Specifies the maximum limit of crack-growth increment.

DAMN – Specifies the minimum limit of crack-growth increment.

SPLO – Specifies spectrum fatigue loading.

KEQV – Specifies the method for the equivalent stress-intensity factor calculation.

KANG – Specifies the method for the kink angle calculation.

Par2 **–**

For *Par1* = MXAN:

Value (in degrees) of the maximum limit of the crack-growth kinking angle. If **CGROW (p. 321)**,SOPT,MXAN,*Par2* is not issued (or *Par2* is unspecified), the program uses a default kinking-angle limit of 20 degrees. *Par2* = 0 forces mode-I crack propagation.

For *Par1* = MNAN:

Cut-off value (in degrees) for the minimum crack-growth kinking angle. Default = 1.

For *Par1* = DAMX:

Value to determine the maximum crack-growth increment. Default: *da*max = 1.5 * *ESIZE*, where *ESIZE* is the reference-element size at the crack-front at any substep.

For *Par1* = DAMN:

Value to determine the minimum crack-growth increment. Default: *da*min = 0.

For *Par1* = SPLO:

WEIG -- Defines the damage weight average method for the crack-growth direction (default).

DMAX -- Defines the dominative load point (maximum damage) method for the crack-growth direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For *Par1* = KEQV:

MTS -- Maximum tangential stress criterion (default).

RICHARD -- Richard approximation function.

POOK -- Pook criterion.

EMPS -- Empirical function based on square root of sum of squares.

For *Par1* = KANG:

MTS -- Maximum tangential stress criterion (default).

RICHARD -- Richard approximation function.

Par3 **–**

For *Par1* = DAMX or DAMN:

ABSO – Indicates that *Par2* is the absolute value (default). The corresponding limit (*da*max or *da*min) is *Par2*.

MULT – Indicates that *Par2* is a multiplier of the reference-element size (*ESIZE*) at the crack-front at any substep. The corresponding limit (*da*max or *da*min) is *Par2* * *ESIZE*.

For information about controlling the element size (*ESIZE*) at the crack-front, see **CGROW (p. 321)**,RMCONT,ESIZE and **CGROW (p. 321)**,RMCONT,CMULT.

For *Par1* = KEQV and *Par2* = RICHARD or EMPS:

α_1– multiplicative factor in $K2$ term in the corresponding function for equivalent SIF.

For *Par1* = KANG and *Par2* = RICHARD:

A – coefficient of first order term in the Richard function for the kink angle.

Par4 **–**

For *Par1* = DAMN:

0 – Sets the crack increments (Δa) less than *da*min (specified via *Par2*) as Δa = *da*min. It is the default option if *Par2* > 0.

1 – Sets the crack increments (Δa) less than *da*min as $\Delta a = 0$.

If *Par2* = 0, the program sets any crack increment (Δa) less than two percent of the reference-element size at the crack-front as $\Delta a = 0$.

For *Par1* = KEQV and *Par2* = RICHARD or EMPS:

α_2– multiplicative factor in $K3$ term in the corresponding function for the equivalent SIF.

For *Par1* = KANG and *Par2* = RICHARD:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B – coefficient of second order term in the Richard function for the kink angle.

Command Specification for *Action* = RMCONT

CGROW (p. 321),RMCONT,*Par1*,*Par2*,*Par3*,*Par4*,*Par5*

Valid in a SMART crack-growth analysis only.

Par1 –

COARSE – Controls mesh-coarsening.

ESIZE – Controls element sizing and crack-increment size.

CMULT – Controls the crack-growth increment multiplier.

Par2 –

For *Par1* = COARSE:

CONS – Remeshing occurs with conservative mesh-coarsening (default).

MODE – Remeshing occurs with moderate mesh-coarsening.

AGGR – Remeshing occurs with aggressive mesh-coarsening.

For *Par1* = ESIZE:

Crack-front reference element-size value, *or* the table name for specifying tabular element size values as a function of solution time.

For *Par1* = CMULT:

Crack-front element-size multiplier function:

TIME – Controls the crack-growth increment multiplier via the beginning and end of the solution time.

CEMX – Controls the crack-growth increment multiplier via the beginning and end of the accumulated maximum crack extension.

RMNU – Controls the crack-growth increment multiplier via the beginning and end of the number of remeshing steps.

TABLE – The name of the table for controlling the crack-growth increment multiplier.

Par3 - Par5 –

For *Par1* = ESIZE and *Par2* = *VALUE* or %*tablename*% and *Par3* = COMP:

Par4 = Name of the node or element component ($\leq$ 32 characters).

For *Par1* = ESIZE and *Par2* = *VALUE* or %*tablename*% and *Par3* = NODE:

Par4 = Node ID.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For *Par1* = ESIZE and *Par2* = *VALUE* or %*tablename*% and *Par3* = ELEM:

Par4 = Element ID.

For *Par1* = CMULT and *Par2* = TIME:

Par3 = *VALUE* – Maximum value of the crack-growth multiplier value.

Par4 = *VALUE* – Time at the beginning of the crack-increment control function.

Par5 = *VALUE* – Time at the end of the crack-increment control function.

For *Par1* = CMULT and *Par2* = CEMX:

Par3 = *VALUE* – Maximum value of the crack-growth multiplier value.

Par4 = *VALUE* – Accumulated maximum crack extension at the beginning of the crack-increment control function.

Par5 = *VALUE* – Accumulated maximum crack extension at the end of the crack-increment control function.

For *Par1* = CMULT and *Par2* = RMNU:

Par3 = *VALUE* – Maximum value of the crack-growth multiplier value.

Par4 = *VALUE* – Remeshing number at the beginning of the crack-increment control function.

Par5 = *VALUE* – Remeshing number at the end of the crack-increment control function.

For *Par1* = CMULT and *Par2* = TABLE:

Par3 = %*tablename*% – Controls the crack-growth increment multiplier via the tabular data in the specified table.

To specify a table (***DIM (p. 530)**), enclose the table name *tablename* within % characters. You can specify one table per crack-growth set, and time is the only valid variable.

Command Specification for *Action* = NPLOAD

CGROW (p. 321),NPLOAD,*Par1*,*Par2*,*Par3*

Valid in a SMART fatigue crack-growth analysis using nonproportional loading.

***Par1* –**

METHOD – Specifies the nonproportional solution method.

DIRECTION – Specifies the nonproportional crack-growth-direction method.

***Par2* –**

For *Par1* = METHOD:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DKEF – Specifies the effective stress-intensity-factor range method.

DKMO – Specifies the stress-intensity-factor range method based on mode separation.

TKEF – Specifies the total effective stress-intensity-factor method.

TKMO – Specifies the total stress-intensity-factor method based on mode separation.

For *Par1* = DIRECTION:

KMAX – Specifies the maximum-load method for the crack-growth direction.

KMIN – Specifies the minimum-load method for the crack-growth direction.

KLOC – Specifies the local kink-tip stress-intensity-factor method.

Par3 **–**

For *Par2* = KLOC:

w – Parameter-fitting value between 0 and 1. Default = 0.5.

Notes

When *Action* = NEW, the **CGROW (p. 321)** command initializes a crack-growth simulation set. Subsequent **CGROW (p. 321)** commands define the parameters necessary for the simulation.

For multiple cracks, issue multiple **CGROW (p. 321)**,NEW commands (and any subsequent **CGROW (p. 321)** commands necessary to define the parameters) for *each crack*.

For SMART-based crack-growth:

- *Action* = CPATH has no effect.
- *Action* = STOP affects both SMART-based static and fatigue crack-growth analyses.
- When *Action* = CSFL with *Option* = CZM, the remeshing for cracks associated with this option is enforced at the end of the first load step and the first substep whether or not the crack grows at this moment. On original crack surfaces (defined via **CINT (p. 334)**,SURF), INTER204 elements are initialized fully damaged, as they are used for crack-closure only and do not contribute bonding tractions on crack surfaces. The cohesive tractions on the new open crack surface are defined via the cohesive zone material model type (**TB (p. 1899)**,CZM) and **CGROW (p. 321)**,CSFL,CZM. This option can be combined with **ADPCI (p. 98)** to initialize a crack with cohesive effect.

For VCCT-based crack-growth:

- Crack-growth element components must use the crack tip nodes as the starting nodes of the crack path.
- Fracture criteria (*Action* = FCOPTION) use energy-release rates calculated via VCCT technology (**CINT (p. 334)**,TYPE,VCCT). For information about the fracture criteria available, see Fracture Criteria in the *Fracture Analysis Guide* or **TB (p. 1899)**,CGCR (p. 1912).

For XFEM-based crack-growth:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The crack specification originates via the **XFENRICH (p. 2193)**, **XFDATA (p. 2192)**, or **XFCRKMESH (p. 2191)** command.
- *Action* = CPATH, DTMIN, or DTMAX has no effect.
- *Action* = STOP affects fatigue crack-growth analysis only.

Menu Paths

This command cannot be accessed from a menu.

CHECK, *Sele*, *Levl*

Checks current database items for completeness.

PREP7 (p. 22): Database (p. 22)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sele

Specifies which elements are to be checked:

(blank)

Check all data.

ESEL

Check only elements in the selected set and unselect any elements not producing geometry check messages. The remaining elements (those producing check messages) can then be displayed and corrected. A null set results if no elements produce a message. Issue **ESEL (p. 661)**,ALL to select all elements before proceeding.

Levl

Used only with *Sele* = ESEL:

WARN

Select elements producing warning and error messages.

ERR

Select only elements producing error messages (default).

Notes

This command will not work if **SHPP (p. 1787)**,OFF has been set. A similar, automatic check of all data is done before the solution begins.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the "Check Elements" option is invoked through the GUI (menu path **Main Menu> Preprocessor> Meshing> Check Elems**), the **CHECK (p. 333)**,ESEL logic is used to highlight elements in the following way: good elements are blue, elements having warnings are yellow, and bad (error) elements are red.

Note:

The currently selected set of elements is not changed by this GUI function.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Meshing>Check Mesh>Individual Elm>Select Warning/Error Elements

---

CHKMSH, *Comp*

Checks area and volume entities for previous meshes.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Comp

Name of component containing areas or volumes.

Notes

CHKMSH (p. 334) invokes a predefined Mechanical APDL macro that checks areas and volumes to find out if they were previously meshed. This macro name will appear in the log file (`Jobname.log`) prior to area and volume meshing operations initiated through the GUI. This command is not intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for use via **/INPUT (p. 948)**).

Menu Paths

This command cannot be accessed from a menu.

CINT, *Action*, *Par1*, *Par2*, *Par3*, *Par4*, *Par5*, *Par6*, *Par7*

Defines parameters associated with fracture-parameter calculations.

PREP7 (p. 22): Special Purpose (p. 37)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Specifies action for defining or manipulating initial crack data:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NEW

Initiate a new calculation and assign an ID. (p. 336)

CTNC

Define the crack-tip node component. (p. 336)

SURF

Define the crack-surface node components. (p. 337)

CENC

Define the crack-extension node component, the crack-tip node, and the crack-extension direction. (p. 337)

TYPE

Define the type of calculation to perform. (p. 337)

DELE

Delete the **CINT (p. 334)** object associated with the specified ID. (p. 339)

NCON

Specify the number of contours to calculate in the contour-integral calculation. (p. 339)

SYMM

Indicate whether the crack is on a symmetrical line or plane. (p. 339)

NORM

Define the crack-plane normal. (p. 340)

UMM

Enable or disable the unstructured-mesh method (UMM). (p. 339)

EDIR

Crack-assist extension direction. (p. 340)

PLOT

Plots the crack-front and crack-tip coordinate system. (p. 341)

LIST

List the **CINT** commands issued, or the elements used (p. 341), in fracture-parameter calculations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CXFE

Define the crack-tip element or crack-front element set. (p. 341) Valid for XFEM-based crack-growth analysis only.

RADIUS

Define the radius at which the given value is to be evaluated. (p. 342) Valid for XFEM-based crack-growth analysis only.

RSWEEP

Define the minimum and maximum sweep angle from existing crack direction. (p. 342) Valid for XFEM-based crack-growth analysis only.

INIT

SMART crack-initiation ID (p. 342).

CSFL

Convert initial-stress data to crack-surface traction loading (p. 342).

Command Specification for *Action* = NEW

CINT (p. 334), NEW, *Par1*

Par1 --

CINT (p. 334) ID number.

Command Specifications for *Action* = CTNC

CINT (p. 334), CTNC, *Par1*, *Par2*, *Par3*

Par1 --

Crack-tip node component name (must be 32 characters or less).

Par2 --

Crack-extension direction calculation-assist node. Any node on the open side of the crack.

Par3 --

Crack front's end-node crack-extension direction override flag:

0 --

Align the extension direction with the edges attached at the two end nodes of the crack front (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 --

Align the extension direction to be perpendicular to the crack front.

Command Specifications for *Action* = SURF

CINT (p. 334), SURF, *Par1*, *Par2*

***Par1* --**

Crack-surface node component 1 (top or bottom crack face). (Component name must be 32 characters or less.)

***Par2* --**

Crack-surface node component 2 (top or bottom crack face, but the *opposite* of *Par1*). (Component name must be 32 characters or less.)

Command Specifications for *Action* = CENC

CINT (p. 334), CENC, *Par1*, *Par2*, *Par3*, *Par4*, *Par5*, *Par6*, *Par7*

***Par1* --**

Crack-extension node component name (**CM (p. 356)**). (Must be 32 characters or less.)

***Par2* --**

Crack-tip node. The crack-tip node defaults to the first node of the crack-extension node component.

***Par3*, *Par4* --**

Coordinate system number (*Par3*) and the number of the axis that is coincident with the crack direction (*Par4*). When these parameters are defined, *Par5*, *Par6* and *Par7* are ignored.

***Par5*, *Par6*, *Par7* --**

Global x, y, and z components of the crack-extension direction vector. (*Par3* and *Par4* must be blank.)

Command Specifications for *Action* = TYPE

CINT (p. 334), TYPE, *Par1*, *Par2*

***Par1* --**

Type of calculation to perform:

JINT --

Calculate J-integral (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SIFS --

Calculate stress-intensity factors.

TSTRESS --

Calculate T-stress.

MFOR --

Calculate material forces.

CSTAR --

Calculate C*-integral.

VCCT --

Calculate energy-release rate using the VCCT method.

PSMAX --

Calculate circumferential stress at the location where $\sigma_{r\theta}$=0 when sweeping around the crack tip at the given radius. Valid in an XFEM-based crack-growth analysis only.

STTMAX --

Calculate maximum circumferential stress when sweeping around the crack tip at the given radius. Valid in an XFEM-based crack-growth analysis only.

Par2 **--**

Auxiliary stress fields and strategy for **3D** stress-intensity factors (*Par1* = SIFS) calculations:

0 --

The plane-strain auxiliary fields are used at the interior nodes along the crack front. The stress-intensity factors at the end nodes of the crack front are set to copy the stress-intensity factors at the adjacent nodes. (Default.)

1 --

The plane-stress auxiliary fields are used over the entire crack front.

2 --

The plane-strain auxiliary fields are used over the entire crack front.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specifications for *Action* = DELE

CINT (p. 334), DELE, *Par1*

Par1 **--**

CINT (p. 334) ID (default = ALL).

Command Specifications for *Action* = NCON

CINT (p. 334), NCON, *Par1*

Par1 **--**

Number of contours to be calculated.

Command Specifications for *Action* = SYMM

CINT (p. 334), SYMM, *Par1*

Par1 **--**

OFF, 0, *or* NO --

No symmetry (default).

ON, 1, *or* YES --

Symmetric about the crack line/plane.

Command Specifications for *Action* = UMM

CINT (p. 334), UMM, *Par1*

Par1 **--**

OFF, 0, *or* NO --

Disable the unstructured-mesh method (UMM) (default).

ON, 1, *or* YES --

Enable the UMM.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specifications for *Action* = NORM

CINT (p. 334), NORM, *Par1*, *Par2*

Par1 --

Coordinate system number (default = 0, global Cartesian).

Par2 --

Axis of coordinate system (default = 2, global Cartesian Y-axis).

Command Specifications for *Action* = EDIR

CINT (p. 334), EDIR, *ITYPE*, *Par2*, *Par3*, *Par4*, *Par5*

ITYPE --

Input type for the crack-assist extension direction. Valid values are CS (coordinate system number) or COMP (component x or y extension direction).

Par2 --

If *ITYPE* = CS, the coordinate system number.

If *ITYPE* = COMP, the x component of the crack-assist extension direction.

Par3 --

If *ITYPE* is CS, the axis representing the crack-assist extension direction.

If *ITYPE* = COMP, the y component of the crack-assist extension direction.

Par4 --

For *ITYPE* = CS, this value is not specified.

For *ITYPE* = COMP, the z component of the crack-assist extension direction.

Par5 --

A reference node on the crack front attached to the crack-assist extension direction. To accurately calculate and flip the crack-extension directions, the crack-assist extension direction defined at this node is rotated as the tangent along the crack front rotates. This capability is useful when the crack-extension directions vary by more than 180 degrees along the crack front.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specifications for *Action* = PLOT

CINT (p. 334), PLOT, *Par1*, *Par2*

Par1 --

Crack ID.

Par2 --

0 -- Disable plotting of crack-tip coordinate system.

1 -- Enable plotting of crack-tip coordinate system (default).

Color codes are white for the crack-extension direction, green for the crack normal, and blue for the direction tangential to the crack front. To clear or delete the plots, issue **/ANNOT (p. 145)**.

Command Specifications for *Action* = LIST

CINT (p. 334), LIST, *CRACKID*, *Par2*, *Par3*, *Par4*

CrackID --

Crack ID. Default = ALL.

Par2 --

No value -- Lists the **CINT (p. 334)** commands issued for the crack. *Par3* and *Par4* are ignored.

ELEM -- Lists the elements used in the fracture-parameter calculation.

Par3 --

Node number on the crack front/tip. Default = ALL. Valid only when *Par2* =ELEM.

Par4 --

Contour number around the crack front/tip. Default = ALL. Valid only when *Par2* =ELEM.

Command Specifications for *Action* = CXFE

CINT (p. 334), CXFE, *Par1*

Par1 --

Crack-tip element number or crack-front component name. (Component name must be 32 characters or less.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specifications for *Action* = RADIUS

CINT (p. 334), RADIUS, *Par1*

Par1 --

Radius at which a value is evaluated (used with **CINT (p. 334)**,TYPE,PSMAX or **CINT (p. 334)**,TYPE,STTMAX only).

Command Specifications for *Action* = RSWEEP

CINT (p. 334), RSWEEP, *Par1*, *Par2*, *Par3*

Par1 --

Number of intervals for the sweep.

Par2 --

Minimum angle of the sweep.

Par3 --

Maximum angle of the sweep

Command Specifications for *Action* = INIT

CINT (p. 334), INIT, *Par1*

Par1 --

ADPCI (p. 98) ID number. The data associated with the **ADPCI (p. 98)** ID is connected to the **CINT (p. 334)** data set to define crack-initiation analysis details (such as crack location and shape, initiation criteria, etc.).

Command Specifications for *Action* = CSFL

CINT (p. 334), CSFL, INIS, STRE

Initial-stress data (mesh-independent) are converted to a traction load acting on the crack-surfaces (specified via **CINT (p. 334)**,SURF). The traction is applied as a step load for all substeps.

The initial-stress data points are specified at various spatial locations in the global Cartesian coordinate system (**CSYS (p. 441)**). The data is required in the vicinity of the crack surfaces only. The program interpolates the initial stress at the centroid of each element face of the crack surfaces, then determines the equivalent traction at the element face based on its orientation.

For more information, see Table 2.2: Material and Load Support for Fracture-Parameter Calculation and Superposition Principle: Initial Stress vs. Crack-Face Traction in the *Fracture Analysis Guide*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Initiate a new calculation via the *Action* = NEW parameter. Subsequent **CINT (p. 334)** commands (with parameters other than NEW) define the input required for the fracture-parameter calculations.

The simplest method is to define crack information via *Action* = CTNC; however, this method limits you to only one node for a given location along the crack front. Use the CTNC option only when all nodes that define the crack front lie in a single plane.

For *Action*= SURF, *Par1* and *Par2* can be the top or bottom crack-face node component. No order is required, provided that if one value the top crach-face node component, the other must be the bottom, and vice-versa. This option is valid only with **CGROW (p. 321)** for crack-growth simulation.

To define crack information at multiple locations along the crack front, use *Action* = CENC. You can issue **CINT (p. 334)**,CENC, *Par1*, etc. multiple times to define the crack-extension node component, the crack tip, and the crack-extension directions at multiple locations along the crack front.

Although you can vary the sequence of your definitions, all specified crack-tip nodes must be at the crack front, and no crack-tip node can be omitted.

You can define the crack-extension direction directly by specifying either *Action* = CENC or *Action* = NORM.

The crack-assist extension direction (*Action* = EDIR) provides a generic extension direction when *Action* = CTNC. It helps to define crack-extension directions based on the connectivity of the crack-front elements. For a 2D case when the crack tangent cannot be calculated, the program uses the provided crack-assist extension direction directly.

For an XFEM-based crack-growth analysis:

- *Action* = CTNC, CENC, NCON, SYMM, UMM, or EDIR have no effect.
- *Action* = CXFE, RADIUS, or RSWEEP are XFEM-specific and invalid for any other type of crack-growth analysis.
- For **CINT (p. 334)**,TYPE, only *Par1* = PSMAX or STTMAX are valid. Other *Par1* values have no effect.

The stress-intensity factors calculation (**CINT (p. 334)**,TYPE,SIFS) applies only to isotropic linear elasticity. Use only one material type for the crack-tip elements that are used for the calculations.

When calculating energy release rates (**CINT (p. 334)**,TYPE,VCCT), do not restrict the results from being written to the database (**/CONFIG (p. 410)**,NOELDB,1) after solution processing; otherwise, incorrect and potentially random results are possible.

Fracture-parameter calculations based on domain integrations such as stress-intensity factors, J-integral, or material force are not supported when contact elements exist inside the domain. The calculations may become path-dependent unless the contact pressure is negligible.

For *Action* = UMM, the default value can be OFF or ON depending on the element type. The **CINT (p. 334)** command overrides the default setting for the given element.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **CINT (p. 334)** command supports only strain data for initial state (**INISTATE (p. 938)**,SET,DTYP,EPEL). Other initial-state capabilities are not supported.

For more information about using the **CINT (p. 334)** command, including supported element types and material behavior, see Calculating Fracture Parameters in the *Fracture Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

CIRCLE, *PCENT*, *RAD*, *PAXIS*, *PZERO*, *ARC*, *NSEG*

Generates circular arc lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

PCENT

Keypoint defining the center of the circle (in the plane of the circle). If *PCENT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

RAD

Radius of the circle. If *RAD* is blank and *PCENT* = P, the radius is the distance from *PCENT* to *PZERO*.

PAXIS

Keypoint defining axis of circle (along with *PCENT*). If *PCENT* = P and *PAXIS* is omitted, the axis is normal to the working plane.

PZERO

Keypoint defining the plane normal to circle (along with *PCENT* and *PAXIS*) and the zero degree location. Need not be in the plane of the circle. This value is not required if *PAXIS* is defined along the Y axis (that is, a circle in the XZ plane).

ARC

Arc length (in degrees). Positive follows right-hand rule about *PCENT*-*PAXIS* vector. Defaults to 360°.

NSEG

Number of lines around circumference (defaults to minimum required for 90°-maximum arcs, that is, 4 for 360°). Number of keypoints generated is *NSEG* for 360° or *NSEG* + 1 for less than 360°.

Notes

Generates circular arc lines (and their corresponding keypoints). Keypoints are generated at regular angular locations (based on a maximum spacing of 90°). Arc lines are generated connecting the keypoints. Keypoint and line numbers are automatically assigned, beginning with the lowest available values

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(**NUMSTR (p. 1301)**). Adjacent lines use a common keypoint. Line shapes are generated as arcs, regardless of the active coordinate system. Line shapes are invariant with coordinate system after they are generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>By Cent & Radius

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>Full Circle

---

CISOL, *n*, *ID*, *node*, *Cont*, *Dtype*

Stores fracture parameter information in a variable.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

n

Arbitrary reference number or name assigned to this variable. Number must be >1 but </= NUMVAR.

ID

Crack ID number.

node

Crack tip node number.

Cont

Contour number.

Dtype

Data type to output:

JINT

J-integral

IIN1

Interaction integral 1

IIN2

Interaction integral 2

IIN3

Interaction integral 3

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

K1

Mode 1 stress-intensity factor

K2

Mode 2 stress-intensity factor

K3

Mode 3 stress-intensity factor

G1

Mode 1 energy release rate

G2

Mode 2 energy release rate

G3

Mode 3 energy release rate

GT

Total energy release rate

MFTX

Total material force X

MFTY

Total material force Y

MFTZ

Total material force Z

CEXT

Crack extension

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CJUMP,*Option*,*Input1*,*Input2*,*Input3*,*Input4*,*Input5*,*Input6*,*Input7*,*Input8*

Initiates a cycle-jump analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Option to be applied:

CRIT

Jump criterion.

INTENT

Declaration of cycle jump analysis intent. For an analysis that begins with a standard solution, specify before the first **SOLVE (p. 1822)** command.

MINCYC

Minimum number of cycles before a jump is allowed (and, if desired, the empirical adjustment of minimum intermediate cycles).

INICYC

Minimum number of initial cycles before a jump is allowed.

MAXJUMP

Maximum allowable jump.

RELTIME

Relative time.

CONTROL

Control-variable selection.

CNMT

Material-ID-dependent jump control.

ADCR

Material-ID- and/or control-variable-dependent jump criteria.

CALC

Jump-calculation option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PERC

Statistical-jump calculation.

OUTP

Diagnostic information output.

Input1

Additional input according to the specified *Option*:

Option	***Input1 =***	***Input2 =***	***Input3, ..., Input8 =***
CRIT	Value of jump criterion.	---	---
INTENT	---	---	---
MINCYC	Minimum number of cycles. Default = 3.	Increase of minimum intermediate cycles. Default = 0.	Decrease of minimum intermediate cycles. Default = 0.
INICYC	Initial number of cycles. Default = 3.	---	---
MAXJUMP	Maximum allowable number of cycles to jump. No default.	---	---
RELTIME	Relative time. Default = time value corresponding to the absolute maximum load for a defined load cycle.	---	---
CONTROL	S – Stress (default).	---	---
	EPPL – Plastic strain.	---	---
	DOF – Degrees of freedom.	---	---
	SVAR – State variables.	MAT	*Input3* = Material ID
		SEL	*Input3*,...,*Input8* = State variable values assigned to six locations.
CNMT	*Input1*,*Input2*,...,*Input6* = Material ID values assigned to six locations.		
ADCR	MAT - Material.	Material ID.	Jump-criterion value.
	CONT - Control variable.	Control-variable label.	
CALC	1 - Slope-based jump calculation (default).	---	---
	2 -Variable-based jump calculation.		
PERC	Percentile value of the cumulative frequency.		
OUTP	---	---	---

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Cycle jump analysis is disabled.

Notes

When *Option* = CONTROL, any input of a control variable replaces the default (S). If stress is also needed in combination with another control variable (or variables), you must specify it explicitly.

CJUMP (p. 347) requires a corresponding cyclic loading analysis (**CLOAD (p. 351)**).

For more information, see Cycle-Jump Analysis in the *Advanced Analysis Guide*.

This command is also valid in PREP7 (**/PREP7 (p. 1479)**).

Menu Paths

This command cannot be accessed from a menu.

/CLABEL, *WN*, *KEY*

Specifies contour labeling.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

KEY

Labeling key:

0 or 1

Label contours with legend or color (default).

-1

No contour labeling.

N

Same as 1 except show alphabetic legend only on every *N*th element.

Command Default

Show contour line labels.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Labels contours for identification with alphabetic legend for vector displays and color for raster displays. Number of contours is automatically reduced to 9 (or fewer) for clarity. Use **/CONTOUR (p. 415)** command to increase (24 maximum for alphabetic labeling; no limit for color labeling).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Labeling

/CLEAR, *Read*

Clears the database.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Read

File read option:

START

Reread `start.ans` file (default).

NOSTART

Do not reread `start.ans` file.

Notes

The **/CLEAR (p. 350)** command resets the database to the conditions present at the beginning of the problem.

The command is typically used between multiple analyses in the same run, or between passes of a multipass analysis (such as between substructure generation, use, and expansion passes).

The command sets the import and Boolean options back to the default, deletes all items from the database, and sets memory values to zero for items derived from database information. (All files remain intact.) The command also resets the jobname to match the currently open session `.log` and `.err` files, returning the jobname to its original value or to the most recent value specified via **/FILNAME (p. 757)** with *KEY* = 1.

After the database is cleared, the `start.ans` file is reread (by default) unless *Read* = NOSTART.

Additional commands cannot be stacked (via the $ separator) on the same line as the **/CLEAR (p. 350)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use caution when placing the **/CLEAR (p. 350)** command within branching constructs (for example, those using ***DO (p. 567)** or ***IF (p. 927)** commands), as the command deletes all parameters including the looping parameter for do-loops. (To preserve your iteration parameter, issue a **PARSAV (p. 1352)** command *prior* to **/CLEAR (p. 350)**, then *follow* **/CLEAR (p. 350)** with a **PARRES (p. 1351)** command.)

This command is valid in any processor. Issuing this command at any point clears the database.

Menu Paths

Utility Menu>File>Clear & Start New

CLOAD, *Option*, *Input1*

Initiates a cyclic-loading analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Option to be applied:

DEFINE

Mark tabular array as a cyclic-loading table.

CYCNUM

Total number of cycles.

CYCTIME

Cycle time.

TSTEP

Enable time-point-range time-stepping.

OUTR

Select time points for output.

MSUB

Multicycle cyclic-loading.

Input1

Additional input according to the specified *Option*:

Option	*Input1* =
DEFINE	BEGIN – Begin marking tabular array as cyclic load table.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	***Input1 =***
	END – End marking tabular array as cyclic load table.
CYCNUM	Total number of cycles.
CYCTIME	Time length of each cycle.
TSTEP	---
OUTR	Set cycle frequency for results. Default = 1.
MSUB	Number of subcycles in a multicycle cyclic loading analysis.

Command Default

Cyclic load analysis is disabled.

Notes

For more information, see Cyclic-Loading Analysis in the *Advanced Analysis Guide*.

This command is also valid in PREP7 (**/PREP7 (p. 1479)**).

Menu Paths

This command cannot be accessed from a menu.

CLOCAL, *KCN, KCS, XL, YL, ZL, THXY, THYZ, THZX, PAR1, PAR2*

Defines a local coordinate system relative to the active coordinate system.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART

Cartesian

1 or CYLIN

Cylindrical (circular or elliptical)

2 or SPHE

Spherical (or spheroidal)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 or TORO

Toroidal

XL, YL, ZL

Location (in the active coordinate system) of the origin of the new coordinate system (R, θ, Z for cylindrical, R, θ,Φ for spherical or toroidal).

THXY

First rotation about local Z (positive X toward Y).

THYZ

Second rotation about local X (positive Y toward Z).

THZX

Third rotation about local Y (positive Z toward X).

PAR1

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local coordinate system by origin location and orientation angles relative to the active coordinate system. This local system becomes the active coordinate system, and is automatically aligned with the active system (that is, x is radial if a cylindrical system is active, etc.). Nonzero rotation angles (degrees) are relative to this automatic rotation. See the **CS (p. 435)**, **CSKP (p. 438)**, **CSWPLA (p. 440)**, and **LOCAL (p. 1059)** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB (p. 1552)** command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CLOG, *IR*, *IA*, - -, - -, *Name*, - -, - -, *FACTA*, *FACTB*

Forms the common log of a variable

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

--,--

Unused fields.

Name

Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--,--

Unused fields.

FACTA

Scaling factor applied to variable *IA* (defaults to 1.0).

FACTB

Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

Notes

Forms the common log of a variable according to the operation:

$$IR = FACTB * \text{LOG}(FACTA \times IA)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Common Log

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/CLOG, *Fname*, *Ext*, --

Copies the session log file to a named file.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path to which the log file is to be copied (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Notes

This command is valid in any processor, but only during an interactive run.

Menu Paths

This command cannot be accessed from a menu.

CLRMSHLN

Clears meshed entities.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

When you use the GUI method to set the number of elements on specified lines, and any of those lines is connected to one or more meshed lines, areas, or volumes, the program gives you the option to clear the meshed entities. This occurs only when you perform this operation via the GUI; the program does not provide such an option when you use the command method (**LESIZE (p. 1041)**).

If you activate the mesh clearing option, the program invokes a Mechanical APDL macro, **CLRMSHLN (p. 355)**, that clears the meshed entities. This macro name will appear in the log file (`Jobname.log`). This macro is for the Mechanical APDL program's internal use only. This command is not intended to be typed in directly in a Mechanical APDL session, although it can be included in an input file for batch input or for use via **/INPUT (p. 948)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

CM,*Cname*,*Entity*,--,*KOPT*

Groups geometry items into a component.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Cname

An alphanumeric name used to identify this component. *Cname* may be up to 256 characters, beginning with a letter and containing only letters, numbers, dots (.) and underscores (_). Component names beginning with an underscore (for example, _LOOP) are reserved for use by Mechanical APDL and should be avoided. Components named "ALL," "STAT," and "DEFA" are not permitted. Overwrites a previously defined name.

Entity

Label identifying the type of geometry items to be grouped:

VOLU

Volumes.

AREA

Areas.

LINE

Lines.

KP

Keypoints.

ELEM

Elements.

NODE

Nodes.

--

Reserved for future use.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KOPT

Controls how element component contents are updated during nonlinear mesh adaptivity analysis:

0 – Component is not updated during remeshing and therefore contains only initial mesh elements (default).

1 – Component is updated during remeshing to contain the updated elements.

This argument is valid only for nonlinear mesh adaptivity analysis with *Entity* = ELEM, and for solid element components only.

Notes

Components may be further grouped into assemblies (**CMGRP (p. 365)**). The selected items of the specified entity type will be stored as the component. Use of this component in the select command (**CMSEL (p. 374)**) causes all these items to be selected at once, for convenience.

A component is a grouping of some geometric entity that can then be conveniently selected or unselected. A component may be redefined by reusing a previous component name. The following entity types may belong to a component: nodes, elements, keypoints, lines, areas, and volumes. A component may contain only 1 entity type, but an individual item of any entity may belong to any number of components. Once defined, the items contained in a component may then be easily selected or unselected (**CMSEL (p. 374)**). Components may be listed (**CMLIST (p. 366)**), modified (**CMMOD (p. 368)**) and deleted (**CMDELE (p. 362)**). Components may also be further grouped into assemblies (**CMGRP (p. 365)**). Other entities associated with the entities in a component (for example, the lines and keypoints associated with areas) may be selected by the **ALLSEL (p. 114)** command.

An item will be deleted from a component if it has been deleted by another operation (see the **KMODIF (p. 985)** command for an example). Components are automatically updated to reflect deletions of one or more of their items. Components are automatically deleted and a warning message is issued if all their items are deleted. Assemblies are also automatically updated to reflect deletions of one or more of their components or subassemblies, but are not deleted if all their components and subassemblies are deleted.

For nonlinear mesh adaptivity analysis, an extra option is available to update the element component contents automatically during the analysis, applicable in cases where the remeshing region overlaps the defined solid element component region. By enabling the option, the component element boundary is maintained, and the validity of the defined component is guaranteed during the entire analysis run; therefore, the component can be used during both solution and postprocessing.

Components are often used as input to other commands. Some commands restrict the component name to 32 characters. For those commands, this limitation is documented within the command description.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Comp/Assembly>Create Component

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CMACEL, *CM_NAME*, *CMACEL_X*, *CMACEL_Y*, *CMACEL_Z*

Specifies the translational acceleration of an element component

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CM_NAME

The name of the element component.

CMACEL_X*, *CMACEL_Y*, *CMACEL_Z

Acceleration of the element component *CM_NAME* in the global Cartesian X, Y, and Z axis directions, respectively.

Notes

The **CMACEL (p. 358)** command specifies the translational acceleration of the element component in each of the global Cartesian (X, Y, and Z) axis directions.

Components for which you want to specify acceleration loading must consist of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components. You cannot apply the loading to an assembly of element components.

To simulate gravity (by using inertial effects), accelerate the structure in the direction opposite to gravity. For example, apply a positive *CMACELY* to simulate gravity acting in the negative Y direction. Units are length/time^2.

You can define the acceleration for the following analyses types:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC), full, VT [1], Krylov [1], or mode-superposition [2] method
- Transient (**ANTYPE (p. 162)**,TRANS), full or mode-superposition [2] method
- Substructure (**ANTYPE (p. 162)**,SUBSTR)

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

Accelerations are combined with the element mass matrices to form a body-force load vector term. Units of acceleration and mass must be consistent to give a product of force units.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **CMACEL (p. 358)** command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *CMACEL_X*, *CMACEL_Y*, and *CMACEL_Z* input values (***DIM (p. 530)**) as a function of both time and frequency for full transient and harmonic analyses.

Related commands for inertia loads are **ACEL (p. 92)**, **CGLOC (p. 319)**, **CGOMGA (p. 320)**, **DCGOMG (p. 501)**, **DOMEGA (p. 571)**, **OMEGA (p. 1324)**, **CMOMEGA (p. 368)**, and **CMDOMEGA (p. 362)**.

See Analysis Tools in the *Theory Reference* for more information.

This command is also valid in **/PREP7 (p. 1479)**.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Gravity>On Components

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Gravity>On Components

---

/**CMAP**,*Fname*,*Ext*,*--*,*Keyword*,*NCNTR*

Changes an existing or creates a new color mapping table.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

If blank, restore color map.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Keyword

Keyword indicating the disposition of the color map file.

(blank)

Loads existing color map file.

CREATE

Starts the CMAP utility and modifies or creates the specified file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SAVE

Writes the active color map to the specified file, which can be imported into future sessions.

NCNTR

Number of contours to be defined. Default = 9 (even if an existing file is being modified). Maximum = 128.

Command Default

Use predefined Mechanical APDL color map table.

Notes

Reads the color map file (RGB index specifications) to change from current specifications. Only one color map may be active at a time.

For 2D drivers (especially Win32c), modifying the color map can produce anomalies, including legend/contour disagreement.

When *Keyword* equals CREATE, the 2D drivers (X11c and Win32c) display the CMAP utility with an additional contour color picker called CONTOURS. Colors selected via the CONTOURS picker affect result contour displays (such as stresses). No other drivers offer the CONTOURS picker in the CMAP utility.

Changing the color map using the **/CMAP (p. 359)** command changes the meaning of the color labels on the **/COLOR (p. 400)** command. See **/COLOR (p. 400)** for other color controls.

This command is valid anywhere.

Menu Paths

Utility Menu>PlotCtrls>Style>Colors>Default Color Map

CMATRIX, *SYMFAC*, *Condname*, *NUMCOND*, *GRNDKEY*, *Capname*

Performs electrostatic field solutions and calculates the self and mutual capacitances between multiple conductors.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SYMFAC

Geometric symmetry factor. Capacitance values are scaled by this factor which represents the fraction of the total device modeled. Defaults to 1.

Condname

Alpha-numeric prefix identifier used in defining named conductor components.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NUMCOND

Total Number of Components. If a ground is modeled, it is to be included as a component. If a ground is not modeled, but infinite elements are used to model the far-field ground, a named component for the far-field ground is not required.

GRNDKEY

Ground key:

0

Ground is one of the components, which is not at infinity.

1

Ground is at infinity (modeled by infinite elements).

Capname

Array name for computed capacitance matrix. Defaults to **CMATRIX (p. 360)**.

Notes

To invoke the **CMATRIX (p. 360)** macro, the exterior nodes of each conductor must be grouped into individual components using the **CM (p. 356)** command. Each set of independent components is assigned a component name with a common prefix followed by the conductor number. A conductor system with a ground must also include the ground nodes as a component. The ground component is numbered last in the component name sequence.

A ground capacitance matrix relates charge to a voltage vector. A ground matrix cannot be applied to a circuit modeler. The lumped capacitance matrix is a combination of lumped "arrangements" of voltage differences between conductors. Use the lumped capacitance terms in a circuit modeler to represent capacitances between conductors.

Enclose all name-strings in single quotes in the **CMATRIX (p. 360)** command line.

See the *Mechanical APDL Theory Reference* for details.

This command does not support multiframe restarts.

Menu Paths

Main Menu>Solution>Solve>Electromagnet>Static Analysis>Capac Matrix

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CMDELE, *Name*

Deletes a component or assembly definition.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

Name of the component or assembly whose definition is to be removed.

Notes

Entities contained in the component, or the components within the assembly, are unaffected. Only the grouping relationships are deleted. Assemblies are automatically updated to reflect deletion of their components or subassemblies, but they are not automatically deleted when all their components or subassemblies are deleted.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Lines

Utility Menu>Select>Comp/Assembly>Delete Comp/Assembly

CMDOMEGA, *CM_NAME*, *DOMEGAX*, *DOMEGAY*, *DOMEGAZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Specifies the rotational acceleration of an element component about a user-defined rotational axis.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

***CM_NAME*,**

The name of the element component.

DOMEGAX*, *DOMEGAY*, *DOMEGAZ

If the *X2*, *Y2*, *Z2* fields are not defined, *DOMEGAX*, *DOMEGAY*, and *DOMEGAZ* specify the components of the rotational acceleration vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *DOMEGAX* is required. *DOMEGAX* specifies the scalar rotational acceleration about the rotational axis. The rotational direction of *DOMEGAX*is designated either positive or negative, and is determined by the "right hand rule."

X1*, *Y1*, *Z1

If the *X2*, *Y2*, *Z2* fields are defined, *X1*, *Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1*, *Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X2, Y2, Z2

The coordinates of the end point of the rotational axis vector.

Notes

Specifies the rotational acceleration components *DOMEGAX*, *DOMEGAY*, and *DOMEGAZ* of an element component *CM_NAME* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point, or a vector connecting two points.

You can define the rotational acceleration and rotational axis with the **CMDOMEGA (p. 362)** command for these analyses:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC) -- full, VT [1], Krylov [1], or mode-superposition [2]
- Transient (**ANTYPE (p. 162)**,TRANS) -- full or mode-superposition [2]
- Substructuring (**ANTYPE (p. 162)**,SUBSTR)
- Modal (**ANTYPE (p. 162)**,MODAL)

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

Rotational velocities are combined with the element mass matrices to form a body force load vector term. Units are radians/time^2.

The **CMDOMEGA (p. 362)** command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *DOMEGAX*, *DOMEGAY*, and *DOMEGAZ* input values (***DIM (p. 530)**) for full transient and harmonic analyses. In this case, if the end point is specified (*X2*, *Y2*, *Z2*), the rotational velocity axis must be along the global X-, Y-, or Z-axis.

Related commands are **ACEL (p. 92)**, **CGLOC (p. 319)**, **CGLOC (p. 319)**, **OMEGA (p. 1324)**, **CMOMEGA (p. 368)**, **DCGOMG (p. 501)**, **DOMEGA (p. 571)**.

See Analysis Tools in the *Theory Reference* for more information.

You can use the **CMDOMEGA (p. 362)** command in conjunction with any one of the following two groups of commands, but not with both groups simultaneously:

GROUP ONE: **OMEGA (p. 1324)**, **DOMEGA (p. 571)**.

GROUP TWO: **CGOMGA (p. 320)**, **DCGOMG (p. 501)**, **CGLOC (p. 319)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Components for which you want to specify rotational loading must consist of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components. You cannot apply the loading to an assembly of element components.

See Acceleration Effect in the *Mechanical APDL Theory Reference* for more information.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By Axis

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By origin

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpt

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpts

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Node

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Accel>On Component

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By Axis

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>By origin

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpt

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Kpts

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Node

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>On Components>Pick Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Accel>On Component

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CMEDIT,*Aname*,*Oper*,*Cnam1*,*Cnam2*,*Cnam3*,*Cnam4*,*Cnam5*,*Cnam6*,*Cnam7*

Edits an existing assembly.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Aname

Name of the assembly to be edited.

Oper

Operation label:

ADD

To add more components. The level of any assembly to be added must be lower than that of the assembly *Aname* (see **CMGRP (p. 365)** command).

DELE

To remove components.

Cnam1, Cnam2, Cnam3, . . . , Cnam7

Names of components and assemblies to be added to or deleted from the assembly.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Comp/Assembly>Edit Assembly

CMGRP,*Aname*,*Cnam1*,*Cnam2*,*Cnam3*,*Cnam4*,*Cnam5*,*Cnam6*,*Cnam7*,*Cnam8*

Groups components and assemblies into an assembly.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Aname

An alphanumeric name used to identify this assembly. *Aname* may be up to 256 characters, beginning with a letter and containing only letters, numbers, dots (.), and underscores (_). Overwrites a previously defined *Aname* (and removes it from higher level assemblies, if any).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cnam1, Cnam2, Cnam3, . . . , Cnam8

Names of existing components or other assemblies to be included in this assembly.

Notes

Groups components and other assemblies into an assembly identified by a name. **CMGRP (p. 365)** is used for the initial definition of an assembly. An assembly is used in the same manner as a component. Up to 5 levels of assemblies within assemblies may be used.

An assembly is a convenient grouping of previously defined components and other assemblies. Assemblies may contain components only, other assemblies, or any combination. A component may belong to any number of assemblies. Up to 5 levels of nested assemblies may be defined. Components and assemblies may be added to or deleted from an existing assembly by the **CMEDIT (p. 365)** command. Once defined, an assembly may be listed, deleted, selected, or unselected using the same commands as for a component. Assemblies are automatically updated to reflect deletions of one or more of their components or lower-level assemblies. Assemblies are not automatically deleted when all their components or subassemblies are deleted.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Comp/Assembly>Create Assembly

CMLIST, *Name*, *Key*, *Entity*

Lists the contents of a component or assembly.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

Name of the component or assembly to be listed (if blank, list all selected components and assemblies). If *Name* is specified, then *Entity* is ignored.

Key

Expansion key:

0

Do not list individual entities in the component.

1 or EXPA

List individual entities in the component.

Entity

If *Name* is blank, then the following entity types can be specified:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VOLU

List the volume components only.

AREA

List the area components only.

LINE

List the line components only.

KP

List the keypoint components only

ELEM

List the element components only.

NODE

List the node components only.

Notes

This command is valid in any processor. For components, it lists the type of geometric entity. For assemblies, it lists the components and/or assemblies that make up the assembly.

Examples of possible usage:

CMLIST (p. 366) - List all selected components.

CMLIST (p. 366), , EXPA - List all selected components and for each component list the underlying entity ID's.

CMLIST (p. 366),*Name* - List the specified component.

CMLIST (p. 366),*Name*,EXPA - List specified component along with all underlying entity ID's.

CMLIST (p. 366), , EXPA,*Entity* - List all selected components of specified entity type. For each component also list the underlying entity ID's.

Menu Paths

Utility Menu>List>Components

Utility Menu>List>Other>Components

Utility Menu>Select>Comp/Assembly>List Comp/Assembly

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CMMOD, *Cname*, *Keyword*, *Value*

Modifies the specification of a component.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Cname

Name of the existing component or assembly to be modified.

Keyword

The label identifying the type of value to be modified.

NAME - Modify the NAME of the component

Value

If *Keyword* is NAME, then the value is the alphanumeric label to be applied. See the **CM (p. 356)** command for naming convention details. If a component named *Value* already exists, the command will be ignored and an error message will be generated.

Notes

The naming conventions for components, as specified in the **CM (p. 356)** command, apply for **CMMOD (p. 368)** (256 characters, "ALL", "STAT" and "DEFA" are not allowed, etc.). However, if you choose a component name that is already designated for another component, an error message will be issued and the command will be ignored.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

CMOMEGA, *CM_NAME*, *OMEGAX*, *OMEGAY*, *OMEGAZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Specifies the rotational velocity of an element component about a user-defined rotational axis.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CM_NAME

The name of the element component.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OMEGAX, OMEGAY, OMEGAZ

If the *X2*, *Y2*, *Z2* fields are not defined, *OMEGAX*, *OMEGAY*, and *OMEGAZ* specify the components of the rotational velocity vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *OMEGAX* is required. *OMEGAX* specifies the scalar rotational velocity about the rotational axis. The rotational direction of *OMEGAX* is designated either positive or negative, and is determined by the "right hand rule."

X1, Y1, Z1

If the *X2*, *Y2*, *Z2* fields are defined,*X1*, *Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1*, *Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

X2, Y2, Z2

The coordinates of the end point of the rotational axis vector.

Notes

Specifies the rotational velocity components *OMEGAX*, *OMEGAY*, and *OMEGAZ* of an element component *CM_NAME* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point or a vector connecting two points.

You can define the rotational velocity and rotational axis for these analysis types:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC) -- full, VT [1], Krylov [1], or mode-superposition [2]
- Transient (**ANTYPE (p. 162)**,TRANS) -- full or mode-superposition [2]
- Substructuring (**ANTYPE (p. 162)**,SUBSTR)
- Modal (**ANTYPE (p. 162)**,MODAL)

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

Rotational velocities are combined with the element mass matrices to form a body-force load vector term. Units are radians/time. Related commands are **ACEL (p. 92)**, **CGLOC (p. 319)**, **CGLOC (p. 319)**, **CGOMGA (p. 320)**, **CMDOMEGA (p. 362)**, **DCGOMG (p. 501)**, **DOMEGA (p. 571)**.

See Analysis Tools in the *Theory Reference* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the **CMOMEGA (p. 368)** command in conjunction with either one of the following two groups of commands, but not with both groups simultaneously:

GROUP ONE: **OMEGA (p. 1324)**, **DOMEGA (p. 571)**.

GROUP TWO: **CGOMGA (p. 320)**, **DCGOMG (p. 501)**, **CGLOC (p. 319)**.

Components for which you want to specify rotational loading must consist of elements only. The elements you use cannot be part of more than one component, and elements that share nodes cannot exist in different element components. You cannot apply the loading to an assembly of element components.

If you have applied the Coriolis effect (**CORIOLIS (p. 417)**) using a stationary reference frame, the **CMOMEGA (p. 368)** command takes the gyroscopic damping matrix into account for the elements listed under *"Stationary Reference Frame"* in the notes section of the **CORIOLIS (p. 417)** command. Mechanical APDL verifies that the rotation vector axis is parallel to the axis of the element; if not, the gyroscopic effect is not applied. If you issue a **CMOMEGA (p. 368)** command when the Coriolis or gyroscopic effect is present, a subsequently issued **OMEGA (p. 1324)** command has no effect.

The **CMOMEGA (p. 368)** command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *OMEGAX*, *OMEGAY*, and *OMEGAZ* input values (***DIM (p. 530)**) for modal, full transient, and full harmonic analyses. In this case, if the end point is specified (*X2*, *Y2*, *Z2*), the rotational velocity axis must be along the global X-, Y-, or Z-axis.

The load interpolation setting (**KBC (p. 970)**) applies to the rotational velocity, in particular the *OMGSQRDKEY* option for quadratic interpolation.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By Axis

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By origin

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpt

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpts

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Node

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Veloc>On Component

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By Axis

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>By origin

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpt

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Kpts

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Node

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>On Components>Pick Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Veloc>On Component

---

CMPLOT,*Label*,*Entity*,*Keyword*

Plots the entities contained in a component or assembly.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Name of the component or assembly to be plotted.

(blank)

All selected components and assemblies are plotted (default). If fewer than 11 components are selected, then all are plotted. If more than 11 components are selected, then only the first 11 are plotted.

ALL

All selected components are plotted. If number of selected components is greater than 11, then the legend showing component names will not be shown.

N

Next set of defined components and assemblies is plotted.

P

Previous set of defined components and assemblies is plotted.

Cname

The specified component or assembly is plotted.

SetNo.

The specified set number is plotted.

Entity

If *Label* is BLANK or ALL, then the following entity types can be specified:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VOLU

Plot the volume components only.

AREA

Plot the area components only.

LINE

Plot the line components only.

KP

Plot the keypoint components only.

ELEM

Plot the element components only.

NODE

Plot the node components only.

Keyword

For *Keyword* = ALL, plot the specified component name in the *Label* field in the context of all entities of the same type. Not valid if *Label* field is BLANK or ALL.

Notes

Components are plotted with their native entities. For assemblies, all native entities for the underlying component types are plotted simultaneously. Although more components can be plotted, the legend displays only 11 at a time. When more than eleven are plotted, the legend is not displayed.

Possible usage:

CMPLOT (p. 371),*CNAME* - Plots the specified component (if selected).

CMPLOT (p. 371),*CNAME*, ALL - Plot component in the context of all other selected entity components of the same type as the component.

CMPLOT (p. 371) - Plot the first eleven selected components.

CMPLOT (p. 371),ALL - Plot all selected components.

CMPLOT (p. 371),N or **CMPLOT (p. 371)**,P - Plot next or previous set of eleven components.

CMPLOT (p. 371),ALL,*Entity* - Plot all selected components of type specified in *Entity*.

CMPLOT (p. 371), ,*Entity* - Plot components of type specified in *Entity*, from the first eleven components.

CMPLOT (p. 371),N,*Entity* - Plot components of type specified in *Entity*, if any, from the next set of eleven components (substitute P for N to plot from previous set).

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>Plot>Components>By Name / Set Number

Utility Menu>Plot>Components>Next Set

Utility Menu>Plot>Components>Previous Set

Utility Menu>Plot>Components>Selected Components

CMROTATE, *CM_Name*, *ROTATX*, *ROTATY*, *ROTATZ*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Specifies the rotational velocity of an element component in a brake-squeal analysis.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CM_Name

The name of the element component.

ROTATX, ROTATY, ROTATZ

If the *X2*, *Y2*, *Z2* fields are not defined, *ROTATX*, *ROTATY*, and *ROTATZ* specify the components of the rotational angle vector in the global Cartesian X, Y, Z directions.

If the *X2*, *Y2*, *Z2* fields are defined, only *ROTATX* is required. *ROTATX* specifies the scalar rotational velocity about the rotational axis. The rotational direction of *ROTATX* is designated either positive or negative, and is determined by the "right hand rule."

X1, Y1, Z1

If the *X2*, *Y2*, *Z2* fields are defined, *X1*, *Y1*, and *Z1* define the coordinates of the beginning point of the rotational axis vector. Otherwise, *X1*, *Y1*, and *Z1* are the coordinates of a point through which the rotational axis passes.

X2, Y2, Z2

The coordinates of the end point of the rotational axis vector.

Notes

The **CMROTATE (p. 373)** command specifies the rotational motion velocity components *ROTATX*, *ROTATY*, and *ROTATZ* of an element component *CM_Name* about a user-defined rotational axis. The rotational axis can be defined either as a vector passing through a single point or a vector connecting two points. **CMROTATE (p. 373)** can be used in static analyses (**ANTYPE (p. 162)**,STATIC) and modal analyses (**ANTYPE (p. 162)**,MODAL).

This command sets the constant rotational velocity on the nodes of the specified element component, despite any deformation at the nodes. This feature is primarily used for generating sliding contact at frictional contact interfaces in a brake-squeal analysis. This type of analysis typically involves surface-to-surface contact between the brake pad and the rotating disk. The applicable contact elements, therefore, are CONTA174 and CONTA175.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A brake-squeal analysis generally involves a linear perturbation modal analysis subsequent to a large-deformation static analysis with the Newton-Raphson option set as **NROPT (p. 1261)**,UNSYM. Therefore, **CMROTATE (p. 373)** is not applicable for multiple load step solves using the **LSSOLVE (p. 1085)** command.

The load interpolation setting (**KBC (p. 970)**) applies to the rotational velocity, in particular the *OMG-SQRDKEY* option for quadratic interpolation.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

CMSEL, *Type*, *Name*, *Entity*

Selects a subset of components and assemblies.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

ALL

Also select all components.

NONE

Unselect all components.

Name

Name of component or assembly whose items are to be selected (valid only if *Type* = S, R, A, or U).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Graphical picking is enabled if *Type* is blank and *Name* = PICK (or simply "P").

Entity

If *Name* is blank, then the following entity types can be specified:

VOLU

Select the volume components only.

AREA

Select the area components only.

LINE

Select the line components only.

KP

Select the keypoint components only.

ELEM

Select the element components only.

NODE

Select the node components only.

Notes

Selecting by component is a convenient adjunct to individual item selection (for example, **VSEL (p. 2163)**, **ESEL (p. 661)**, etc.). **CMSEL (p. 374)**, ALL allows you to select components **in addition** to other items you have already selected.

If *Type* = R for an assembly selection (**CMSEL (p. 374)**,R,<*assembly-name*>), the reselect operation is performed on each component in the assembly in the order in which the components make up the assembly. Thus, if one reselect operation results in an empty set, subsequent operations will also result in empty sets. For example, if the first reselect operation tries to reselect node 1 from the selected set of nodes 3, 4, and 5, the operation results in an empty set (that is, no nodes are selected). Since the current set is now an empty set, if the second reselect operation tries to reselect any nodes, the second operation also results in an empty set, and so on. This is equivalent to repeating the command **CMSEL (p. 374)**,R,<*component-name*> once for each component making up the assembly.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Joint Elems

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Joint Elems

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Joint Elems

Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Joint Elems

Utility Menu>Select>Comp/Assembly>Select Comp/Assembly

CMSFILE,*Option*,*Fname*,*Ext*,*CmsKey*

Specifies a list of component mode synthesis (CMS) results files for plotting results on the assembly.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Specifies the command operation:

ADD

Add the specified component results file (*Fname*) to the list of files to plot. This option is the default.

DELETE

Remove the specified component results file (*Fname*) from the list of files to plot.

LIST

List all specified component results files.

CLEAR

Clear all previous files added.

ALL

Add *all* component results (`.rst`) files from the working directory to the list of files to plot.

Fname

The file name (with full directory path) of the component results file. The default file name is the `Jobname` (specified via the **/FILNAME (p. 757)** command).

Ext

The file name (*Fname*) extension. The default extension is `.rst`.

CmsKey

Valid only when adding a results file (*Option* = ADD or ALL), this key specifies whether or not to check the specified `.rst` file to determine if it was created via a CMS expansion pass:

ON

Check (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Do not check.

Command Default

If issued with no arguments, the **CMSFILE (p. 376)** command uses these defaults:

CMSFILE (p. 376),ADD,*Jobname*,rst,ON

The command adds the component results file `Jobname.rst`.

Notes

The **CMSFILE (p. 376)** command specifies the list of component mode synthesis (CMS) results files to include when plotting the mode shape of an assembly.

During postprocessing (**/POST1 (p. 1461)**) of a CMS analysis, issue the **CMSFILE (p. 376)** command to point to component results files of interest. (You can issue the command as often as needed to include all or some of the component results files.) Issue the **SET (p. 1724)** command to acquire the frequencies and mode shapes from substeps for all specified results files. Execute a plot (**PLNSOL (p. 1425)**) or print (**PRNSOL (p. 1505)**) operation to display the mode shape of the entire assembly.

When you specify a results file to add to the plot list, the default behavior of the command (*CmsKey* = ON) is to first verify that the file is from a CMS analysis and that the frequencies of the result sets on the file match the frequencies on the first file in the list. If *CmsKey* = OFF, you can add any `.rst` file to the list of files to plot, even if the file was not expanded via a CMS expansion pass.

If *CmsKey* = ON (default), output from the command appears as: `ADD CMS FILE = filename.rst`. If *CmsKey* = OFF, output from the command appears as: `ADD FILE = filename.rst`.

If *Option* = DELETE or CLEAR, you must clear the database (**/CLEAR (p. 350)**), then re-enter the post-processor (**/POST1 (p. 1461)**) and issue a **SET (p. 1724)** command for the change to take effect on subsequent plots.

Clearing the database does *not* clear the list of files specified via the **CMSFILE (p. 376)** command. Specify *Option* = CLEAR to clear the list of files.

Menu Paths

Main Menu>General Postproc>Data & File Opts

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CMSOPT,*Cmsmeth*,*NMODE*,*FREQB*,*FREQE*,*Fbddef*,*FBDVAL*,*IOkey*,--,--,*Elcalc*,--,*Eigmeth*,*nStartVN*

Specifies component mode synthesis (CMS) analysis options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 381)

Cmsmeth

The component mode synthesis method to use. This value is required.

FIX

Fixed-interface method.

FREE

Free-interface method.

RFFB

Residual-flexible free-interface method.

NMODE

The number of normal modes extracted and used in the superelement generation. This value is required; the minimum is 1.

FREQB

Beginning, or lower end, of frequency range of interest. This value is optional.

The program always sets this value to zero if the residual-flexible free-interface method (*Cmsmeth* = RFFB) or the free-interface method (*Cmsmeth* = FREE) is specified via **RESVEC (p. 1603)**.

FREQE

Ending, or upper end, of frequency range of interest. This value is optional.

Fbddef

In a free-interface (*Cmsmeth* = FREE) or residual-flexible free-interface (*Cmsmeth* = RFFB) CMS analysis, the method to use for defining free body modes:

FNUM

The number (*FDBVAL*) of rigid body modes in the calculation.

FTOL

Employ a specified tolerance (*FDBVAL*) to determine rigid body modes in the calculation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FAUTO

Automatically determine rigid body modes in the calculation. This method is the default.

RIGID

If no rigid body modes exist, define your own via the **RIGID (p. 1609)** command.

FBDVAL

In a free-interface CMS analysis (*Cmsmeth* = FREE), the number of rigid body modes if *Fbddef* = FNUM (where the value is an integer from 0 through 6), *or* the tolerance to employ if *Fbddef* = FTOL (where the value is a positive real number representing rad/sec). This value is required only when *Fbddef* = FNUM or *Fbddef* = FTOL; otherwise, any specified value is ignored.

IOkey

Output key to control writing of the complete transformation matrix on the `.cms` file to the `.tcms` file (FIX or FREE methods) or body properties to the `.EXB` file (FIX method).

CMS

Write the complete transformation matrix of the nodal component on the `.cms` file. For more information, see CMS File Format in the *Programmer's Reference*.

TCMS

Write the transformation matrix of the nodal component defined via **OUTPR (p. 1332)** to a `.tcms` file. For more information, see TCMS File Format in the *Programmer's Reference*.

EXB

Write a body property input file (`.EXB` file) containing the condensed substructure matrices and other body properties for use with AVL EXCITE. For more information, see Ansys Interface to AVL EXCITE in the *Substructuring Analysis Guide*.

--,--

Unused fields.

Elcalc

Element calculation key:

NO

Do not calculate element results (default).

YES

Calculate element results and write them to the `.cms` file for the expansion pass.

--

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eigmeth

Mode extraction method to be used for the symmetric eigenvalue problem during the generation pass:

LANB

Block Lanczos algorithm (default).

SUBS

Subspace algorithm.

SNODE

Supernode algorithm.

nStartVN

Node number to be assigned to the first virtual node created to store the generalized coordinates. See "Notes" (p. 380) for more information.

Command Default

Issuing the **CMSOPT (p. 378)** command with no arguments is invalid. You must specify at least the CMS method (*Cmsmeth*) and the number of modes (*NMODE*). In a free-interface (*Cmsmeth* = FREE) or residual-flexible free-interface (*Cmsmeth* = RFFB) CMS analysis, the default method for determining rigid body modes is FAUTO (automatic).

Notes

CMS supports damping matrix reduction when a damping matrix exists. Set the matrix generation key to 3 (**SEOPT (p. 1715)**,*Sename*,*SEMATR*) to generate and then reduce stiffness, mass, and damping matrices.

CMS does not support the **SEOPT (p. 1715)**,,,,,RESOLVE command. Instead, the program sets the expansion method for the expansion pass (*EXPMTH*) to BACKSUB.

By default, the static constraint modes are not written to the `.cms` file for the fixed-interface and free-interface methods. Issue *IOkey* = CMS to write them.

If *IOkey* = TCMS, the transformation matrix is printed out and written to the `.tcms` file when the **OUTPR (p. 1332)** command is issued with *ITEM* = NSOL and *FREQ* not equal to NONE. In addition, the transformation matrix is printed out when *SEPR* is equal to 1 or 2 on **SEOPT (p. 1715)**. In interactive sessions, the transformation matrix is not output if the model has more than 10 elements.

For information about the component modes stored in the `.cms` or `.tcms` file, refer to Component Modes Storage.

If *Elcalc* = YES, the element results of the component modes included in the transformation matrix of the CMS method are calculated and written to the `.cms` file. This can significantly reduce the computation time of the The CMS Use and Expansion Passes in the *Substructuring Analysis Guide*. For limit-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ations and available element results, see Calculating the Element Results in the *Substructuring Analysis Guide*.

Select a *nStartVN* value to offset the virtual node numbers from the other node numbers used in the model; otherwise, the program selects *nStartVN* to fulfill that condition. In the case of multiple superelements, if *nStartVN* is defined during each generation pass, then in the use pass, the virtual nodes of all imported superelements are gathered and renumbered from the *nStartVN* value specified for the first encountered superelement (first **SE (p. 1651)** command). *nStartVN* can also be defined in the use pass via **SE (p. 1651)**. (If *nStartVN* is defined by both the **CMSOPT (p. 378)** and **SE (p. 1651)** commands, the larger number prevails.)

For more information, see Component Mode Synthesis in the *Substructuring Analysis Guide*.

This command is also valid in **/PREP7 (p. 1479)**.

Product Restrictions

Ansys Mechanical Enterprise PrepPost

Only **CMSOPT (p. 378)**,FIX,,,,,,EXB is supported.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

CMWRITE, *Fname*, *Ext*, --, --, *Fmat*

Writes node and element components and assemblies to a file.

DATABASE (p. 13): Components (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to CM if *Fname* is blank.

--

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fmat

Format of the output file (defaults to BLOCKED).

BLOCKED

Blocked format. This format allows faster reading of the file.

UNBLOCKED

Unblocked format.

Menu Paths

This command cannot be accessed from a menu.

CNCHECK, *Option*, *RID1*, *RID2*, *RINC*, *InterType*, *TRlevel*, *Val1*, *Val2*, *Val3*

Provides and/or adjusts the initial status of contact pairs.

PREP7 (p. 22): Database (p. 22)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Option to be performed:

DETAIL

List all contact pair properties (default).

SUMMARY

List only the open/closed status for each contact pair.

POST

Execute a partial solution to write the initial contact configuration to the `jobname.rcn` file.

ADJUST

Physically move contact nodes to the target in order to close a gap or reduce penetration. The initial adjustment is converted to structural displacement values (UX, UY, UZ) and stored in the `jobname.rcn` file.

MORPH

Physically move contact nodes to the target in order to close a gap or reduce penetration, and also morph the underlying solid mesh. The initial adjustment of contact nodes and repositioning of solid element nodes due to mesh morphing are converted to structural displacement values (UX, UY, UZ) and stored in the `jobname.rcn` file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TADJUST

Physically move target body to the contact surface in order to close a gap or reduce penetration. The initial adjustment is converted to structural displacement values (UX, UY, UZ) and stored in the `jobname.rcn` file.

RESET

Reset target element and contact element key options and real constants to their default values. This option is not valid for general contact.

AUTO

Automatically sets certain real constants and key options to recommended values or settings in order to achieve better convergence based on overall contact pair behaviors. This option is not valid for general contact.

TRIM

Trim contact pair (remove certain contact and target elements).

OVER

When a rigid beam or a weld joint connects to a rigid surface constraint or a rigid body, automatically convert the rigid beam elements (MPC184) to rigid target or contact elements.

In addition, if *Val1* = 1 is set, when a Lagrange-multiplier-based joint element connects a rigid surface constraint or a rigid body, also switch to the penalty method (KEYOPT(2) = 1 of MPC184).

UNSE

Unselect certain contact and target elements.

SPLIT

Split base (original) contact pairs into smaller sub-pairs at the preprocessing (**/PREP7 (p. 1479)**) level. The main intent of this option is to achieve better scalability in a distributed-memory parallel (DMP) run. The splitting operation may create additional overlapping contact elements at the split boundaries. Contact pairs can only be split once. Repeated use of this option results in no further splitting for those contact pairs already split.

DMP

This option is similar to the SPLIT, but it is more automatic and contact pair-splitting is done at the solution level (**SOLVE (p. 1822)**) of the first load step, not at the preprocessing level. This option is activated only in a distributed-memory parallel (DMP) run. For this option, *TRlevel*, *InterType*, and *Val1* are valid. All other arguments are ignored.

MERGE

Merge all contact sub-pairs that were previously split (by prior *Option* = SPLIT or DMP operations) back to their original pairs. Any contact and target elements deleted due to

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the trim logic of the splitting operation cannot be recovered by the MERGE operation. All other arguments are ignored.

RID1, *RID2*, *RINC*

The meanings of *RID1*, *RID2*, and *RINC* vary depending on the contact type or the *Option* specified, as described below. *RID1* accepts tabular input for some *Option* values. *RID1*, *RID2*, *RINC* are ignored when *Option* = DMP.

Pair-Based Contact

For pair-based contact, the range of real constant pair IDs for which *Option* will be performed. If *RID2* is not specified, it defaults to *RID1*. If no value is specified, all contact pairs in the selected set of elements are considered.

General Contact

For general contact (*InterType* = GCN), *RID1* and *RID2* are section IDs associated with general contact surfaces instead of real constant IDs. If *RINC* = 0, the *Option* is performed between the two sections, *RID1* and *RID2*. If *RINC* > 0, the *Option* is performed among all specified sections (*RID1* to *RID2* with increment of *RINC*).

Contact Splitting

For contact splitting at the preprocessing level (*Option* = SPLIT only), *RID1*, *RID2*, and *RINC* are used as follows:

- If *RID1*, *RID2*, and *RINC* are non-zero and positive, split the contact pairs from real constant pair ID number *RID1* to *RID2* in increments of *RINC*. In this case, if *TRlevel* is non-zero, it will only be applied to these specified pairs.
- If *RID1* is zero or blank, *TRlevel* will take affect for all contact pairs in the model from largest to smallest.
- If *RID1*, *RID2*, *RINC*, and *TRlevel* are not defined, the program automatically determines the number of sub-pairs for splitting each contact pair.

Tabular Input for *RID1*

RID1 accepts tabular input in the form of a 2D array when *Option* = ADJUST, MORPH or TADJUST. In this case, *RID2*, *RINC*, *Val1*, *Val2*, and *Val3* are ignored. For more information, see Physically Moving Contact Nodes Toward the Target Surface in the *Contact Technology Guide*.

InterType

The type of contact interface (pair-based versus general contact) to be considered; or the type of contact pair to be trimmed, unselected, or auto-set.

The following labels specify the type of contact interface:

(blank)

Include all contact definitions (pair-based and general contact).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GCN

Include general contact definitions only (not valid when *Option* = RESET or AUTO).

The following labels specify the type of contact pairs to be trimmed/unselected/auto-set (used only when *Option* = TRIM, UNSE, or AUTO, and only for pair-based contact definitions):

ANY

All types (default).

MPC

MPC-based contact pairs (KEYOPT(2) = 2).

BOND

Bonded contact pairs (KEYOPT(12) = 3, 5, 6).

NOSP

No separation contact pairs (KEYOPT(12) = 2, 4).

SMAL

Small sliding contact pairs (KEYOPT(18) = 1).

SELF

Self contact pairs (target surface completely overlaps contact surface).

INAC

Inactive contact pairs (symmetric contact pairs for MPC contact or KEYOPT(8) = 2).

The following labels specify the type of contact pairs to be split, and are used only when *Option* = DMP or SPLIT, and only for pair-based contact definitions:

(blank)

All types.

MPC

MPC-based contact pairs (KEYOPT(2) = 2).

BOND

Bonded contact pairs (KEYOPT(12) = 3, 5, 6).

NOSP

No separation contact pairs (KEYOPT(12) = 2, 4).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SMAL

Small sliding contact pairs (KEYOPT(18) = 1).

TRlevel

This argument is either the trimming level for trimming contact pairs, or the number of sub-pairs for contact splitting or executing a partial solution.

Trimming level (used only when *Option* = TRIM, UNSE, or MORPH):

(blank)

Normal trimming (default): remove/unselect contact and target elements which are in far-field.

AGGRE

Aggressive trimming: remove/unselect contact and target elements which are in far-field, and certain elements in near-field.

Number of sub-pairs used for contact splitting (used only when *Option* = SPLIT or DMP):

(blank)

The program automatically chooses the number of sub-pairs for splitting in order to achieve better scalability in a DMP run.

N

Input a non-zero positive number to indicate the maximum number of sub-pairs for splitting the largest contact pair in the model. All other smaller contact pairs will be split following this number proportionally. The number you input may not always be honored. Splitting may result in a fewer number of sub-pairs basing on many factors.

Effect of initial interface treatment used for or executing a partial solution (when *Option* = POST):

(blank)

The program follows settings of KEYOPT(9) of contact element type for every contact pair to report numerical penetration/gap.

1

The program ignores settings of KEYOPT(9) of contact element type for every contact pair to report geometrical penetration/gap.

The remaining arguments, *Val1*, *Val2*, and *Val3*, are parameters that control initial contact adjustment. They are only valid when *Option* = ADJUST, MORPH, TADJUST, OVER, DMP, or SPLIT. The meaning of these arguments varies depending on the *Option* setting.

When *Option* = ADJUST or MORPH, *Val1*, *Val2*, and *Val3* are *CGAP*, *CPEN*, and *IOFF*, respectively:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CGAP

Control parameter for opening gap; must be greater than zero. Close the opening gap if the value of the gap is smaller than the *CGAP* value. *CGAP* defaults to 0.25*PINB (where PINB is the pinball radius) for bonded and no-separation contact. Otherwise, it defaults to the value of real constant ICONT.

CPEN

Control parameter for initial penetration. *CPEN* must be greater than zero. Close the initial penetration if the value of the penetration is smaller than the *CPEN* value. *CPEN* defaults to 0.25*PINB (where PINB is the pinball radius) for any type of interface behavior (either bonded or standard contact).

IOFF

Control parameter for initial adjustment. Input a positive value to adjust the contact nodes towards the target surface with a constant interference distance equal to *IOFF*. Input a negative value to adjust the contact node towards the target surface with a uniform gap distance equal to the absolute value of *IOFF*.

When *Option* = TADJUST, *Val1* and *Val2* are *PMAX* and *PMIN*, respectively (*Val3* is not used):

PMAX

Upper limit of initial allowable penetration.

PMIN

Lower limit of initial allowable penetration.

When *Option* = SPLIT or DMP, *Val1* is the label that specifies the type of contact pairs to be trimmed (only for pair-based contact definitions):

(blank)

The program automatically deletes inactive contact pairs defined by auto-asymmetric selection (KEYOPT(8) = 2). If *Option* = DMP, the program also trims split contact pairs that are associated with bonded contact (KEYOPT(12) = 5 or 6) or small sliding contact (KEYOPT(18) = 1).

TRIM

The program automatically deletes inactive contact pairs defined by auto-asymmetric selection (KEYOPT(8) = 2), and also trims all split contact pairs.

Notes

The **CNCHECK (p. 382)** command provides information for surface-to-surface, node-to-surface, and line-to-line contact pairs (element types TARGE169, TARGE170, CONTA172, CONTA174, CONTA175, CONTA177). All contact and target elements of interest, along with the solid elements and nodes attached to them, must be selected for the command to function properly. For performance reasons, the program uses a subset of nodes and elements based on the specified contact regions (*RID1*, *RID2*, *RINC*) when executing the **CNCHECK (p. 382)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CNCHECK (p. 382) is available in both the PREP7 and SOLUTION processors, but only before the first solve operation (that is, only before the first load step or the first substep).

If the contact and target elements were generated through mesh commands (**AMESH (p. 121)**, **LMESH (p. 1057)**, etc.) instead of the **ESURF (p. 682)** command, you must issue **MODMSH (p. 1138)**,DETACH before **CNCHECK (p. 382)**. Otherwise, **CNCHECK (p. 382)** will not work correctly.

The following additional notes are available:

Option = POST

Option = ADJUST or MORPH

Option = TADJUST

Option = RESET

Option = AUTO

Removing or Unselecting Contact and Target Elements (Option = TRIM/UNSEL)

Option = SPLIT, DMP, or MERGE

Option = POST

The command **CNCHECK (p. 382)**,POST solves the initial contact configuration in one substep. After issuing this command, you can postprocess the contact result items as you would for any other converged load step. However, only the contact status, contact penetration or gap, and contact pressure will have meaningful values. Other contact quantities (friction stress, sliding distance, chattering) will be available but are not useful.

Initial contact results obtained and listed from **CNCHECK (p. 382)**,DETAIL, **CNCHECK (p. 382)**,POST, and the actual solution may not be always consistent with each other. The discrepancy is mainly due to applied boundary conditions and solution options. Contact results from **CNCHECK (p. 382)**,POST are, in general, closer to what the solution will provide than those from **CNCHECK (p. 382)**,DETAIL.

To inspect the real geometric penetration and gap, specify *TRLEVEL*=1. The program will set KEYOPT(9) = 0 internally during the execution of **CNCHECK (p. 382)**,POST.

Because *Option* = POST forces a solve operation, the PrepPost (PP) license (p. 2) does not work with **CNCHECK (p. 382)**,POST.

If **CNCHECK (p. 382)**,POST is issued within the solution processor, the **SOLVE (p. 1822)** command that solves the first load step of your analysis should appear in a different step, as shown in the following example:

```
/SOLU
CNCHECK,POST
FINISH
. . .

/SOLU
SOLVE
FINISH
. . .
```

CNCHECK (p. 382),POST writes initial contact results to a file named `jobname.rcn`. The command should be issued *before* defining analysis type, solution options, and boundary conditions. If the command is issued later, a segmentation violation error may occur. When postprocessing the initial contact state,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you need to explicitly read results from this file using the **FILE (p. 752)** and **SET (p. 1724)**,FIRST commands in POST1 to properly read the corresponding contact data. Otherwise, the results may be read improperly. The following example shows a valid command sequence for plotting the initial contact gap:

```
/SOLU
CNCHECK,POST
FINISH
/POST1
FILE,Jobname,RCN
SET,FIRST
PLNSOL,CONT,GAP,0,1
FINISH
. . .
```

Option = ADJUST or MORPH

You can issue **CNCHECK (p. 382)**,ADJUST to physically move contact nodes to the target surface. Alternatively, you can issue **CNCHECK (p. 382)**,MORPH to physically move contact nodes to the target surface and then morph the underlying mesh to improve the mesh quality. See Physically Moving Contact Nodes Toward the Target Surface in the *Contact Technology Guide* for more information. Similar to the POST option, if **CNCHECK (p. 382)**,ADJUST or **CNCHECK (p. 382)**,MORPH is issued within the solution processor, the **SOLVE (p. 1822)** command that solves the first load step of your analysis should appear in a different step:

```
/SOLU
CNCHECK,ADJUST
FINISH
. . .

/SOLU
SOLVE
FINISH
. . .
```

After issuing the **CNCHECK (p. 382)**,ADJUST command, the initial adjustment is converted to structural displacement values (UX, UY, UZ) and stored in a file named `jobname.rcn`. Similarly, the **CNCHECK (p. 382)**,MORPH command converts the initial adjustment of contact nodes as well as the morphing adjustment of solid element nodes to structural displacement values (UX, UY, UZ) and stores them in the `jobname.rcn` file. You can use this file to plot or list nodal adjustment vectors or create a contour plot of the adjustment magnitudes via the displacements. When postprocessing the nodal adjustment values, you need to explicitly read results from this file using the **FILE (p. 752)** and **SET (p. 1724)**,FIRST commands in POST1 to properly read the corresponding contact data. Otherwise, the results may be read improperly.

Note:

The `jobname.rcn` file contains information generated from the **CNCHECK (p. 382)**,POST, **CNCHECK (p. 382)**,ADJUST, **CNCHECK (p. 382)**,MORPH, or **CNCHECK (p. 382)**, TADJUST command. If multiple commands are issued in the same analysis, the file is overwritten by the last **CNCHECK (p. 382)** command.

Option = TADJUST

You can issue **CNCHECK (p. 382)**,TADJUST to physically move the target body to the contact surface. This command tries to establish initial contact with penetration in a range specified by *PMAX* and *PMIN*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Similar to the ADJUST and MORPH options, the initial adjustment is converted to structural displacement values (UX, UY, UZ) and stored in the `jobname.rcn` file. This option accepts tabular input in the *RID1* field.

You can specify either a positive or negative value for *PMIN* and *PMAX*. The program interprets a positive value as a scaling factor and interprets a negative value as the absolute value.

For more information, see Physically Moving the Target Body Toward the Contact Surface in the *Contact Technology Guide*.

Option = RESET

The command **CNCHECK (p. 382)**,RESET allows you to reset all but a few key options and real constants associated with the specified contact pairs (*RID1*, *RID2*, *RINC*) to their default values. This option is only valid for pair-based contact definitions.

The following key options and real constants remain unchanged when this command is issued:

Element type	Key options not affected by RESET	Real constants not affected by RESET
TARGE169, TARGE170	KEYOPT(2), KEYOPT(3)	R1, R2
CONTA172	KEYOPT(1), KEYOPT(3)	R1, R2
CONTA174, CONTA175, CONTA177	KEYOPT(1)	R1, R2

Option = AUTO

The command **CNCHECK (p. 382)**,AUTO automatically changes certain default or undefined key options and real constants to optimized settings or values. The changes are based on overall contact pair behaviors. In general, this command improves convergence for nonlinear contact analysis. This option is only valid for pair-based contact definitions.

The tables below list typical KEYOPT and real constant settings implemented by **CNCHECK (p. 382)**,AUTO. The actual settings implemented for your specific model may vary from what is described here. You should always verify the modified settings by issuing **CNCHECK (p. 382)**,DETAIL to list current contact pair properties.

KEYOPT	Description	Default (0 or blank)	CNCHECK,AUTO
1	Selects DOF set	Structural DOFs	Automatic selection based on DOFs of underlying elements.
2	Contact algorithm	Augmented Lagrange	1- Penalty for rigid-rigid contact [1].
4	Location of contact detection point	Gauss point	2 - Normal to target surface if KEYOPT(2) > 1.
5	CNOF/ICONT adjustment	No adjustment	1 - Auto CNOF if tiny gap exists.
6	Contact stiffness variation	Use default range	1 - Nominal refinement for opening contact or underlying elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			having **TB (p. 1899)** plasticity. *or* 2 - Aggressive refinement for opening contact and underlying elements having **TB (p. 1899)** plasticity.
7	Element level time increment control	No control	1 - Automatic bisection for self contact. *or* 4 - Impact constraint for opening contact and transient analyses.
8	Assymetric contact selection	No action	2 - Auto selection if KEYOPT(2) > 1.
9	Effect of initial penetration or gap	Include all	1 - Exclude if KEYOPT(5) = 1, or if ICONT was previously specified.
10	Contact stiffness update	Between load steps	0 - Between iterations except when underlying elements are superelements. [2]

1. Set to 0 if KEYOPT(2) > 1 for debonding.

2. Set to 1 if underlying elements are superelements, or if KEYOPT(9) = 2 was previously specified.

Real Constants		Description	Default	CNCHECK,AUTO
No.	Name			
3	FKN	Normal penalty stiffness factor	1	Set to 5 if KEYOPT(9) = 2 (ramp initial penetration) and KEYOPT(10) > 0.
6	PINB	Pinball region	see [1]	Cut in half if spurious contact is detected or contact searching is slow.
14	TCC	Thermal contact conductance	0	Calculated as a function of highest conductivity of underlying element and overall model size.
19	ECC	Electric contact conductance	0	Calculated as a function of highest permitivity or lowest resistivity of underlying element and overall model size.
26	MCC	Magnetic contact permeance	0	Calculated as a function of highest emissivity of underlying element and overall model size.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37	PCC	Pore fluid contact permeability coefficient	0	Calculated as a function of highest permitivity of underlying element and overall model size.

1. PINB default depends on contact behavior (rigid vs. flexible target), **NLGEOM (p. 1231)**,ON or OFF, KEYOPT(9) setting, KEYOPT(12) setting, and the value of real constant CNOF (see Defining the Pinball Region (PINB)).

CNCHECK (p. 382),AUTO also sets **PRED (p. 1476)**,OFF for the case of a force-distributed constraint defined via MPC contact.

Removing or Unselecting Contact and Target Elements (*Option* = TRIM/UNSEL)

You can issue **CNCHECK (p. 382)**,TRIM or **CNCHECK (p. 382)**,UNSEL to remove or unselect contact and target elements which are in far-field (that is, open and not near contact), or even in near-field if aggressive trimming logic is used (*TRlevel* = AGGRE).

Option = SPLIT, DMP, or MERGE

The SPLIT, DMP, and MERGE options are intended for use in a distributed-memory parallel (DMP) run. Use the command **CNCHECK (p. 382)**,SPLIT or **CNCHECK (p. 382)**,DMP to split large contact pairs into smaller sub-pairs so that the smaller contact pairs can be distributed into different cores, thereby improving performance. This functionality is not the same as manually splitting contact pairs during the modeling process. The program splitting logic performs the necessary exchange of data on the boundaries between the split contact pairs to ensure that the results with and without splitting are essentially identical.

The command **CNCHECK (p. 382)**,MERGE can be used to merge the sub-pairs together again, which is useful for postprocessing the contact results based on the original contact pair geometry. However, caution must be taken when a downstream analysis is performed since the MERGE operation may alter the database.

For more information, see Solving Large Contact Models in a Distributed-Memory Parallel Environment in the *Contact Technology Guide*.

Menu Paths

Main Menu> Preprocessor> Modeling> Create> Contact Pair

CNKMOD, *ITYPE*, *KNUM*, *VALUE*

Modifies contact element key options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITYPE

Contact element type number as defined on the **ET (p. 686)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KNUM

Number of the KEYOPT to be modified (KEYOPT(*KNUM*)).

VALUE

Value to be assigned to the KEYOPT.

Notes

The **CNKMOD (p. 392)** command has the same syntax as the **KEYOPT (p. 979)** command. However, it is valid only in the SOLUTION processor. The command can be used to modify certain contact element KEYOPT values between load steps in any analysis, including restarts and linear perturbation analyses, as shown in the table below.

KEYOPT	Valid in Linear Perturbation	Valid between load steps in other analyses (including restart)
KEYOPT(3)	YES	NO
KEYOPT(12)	YES	YES
KEYOPT(15)	NO	YES

In a multiframe restart, **CNKMOD (p. 392)** must be issued again during each subsequent restart run.

Modifying KEYOPT(12)

A common use for the **CNKMOD (p. 392)** command is to modify contact interface behavior between load steps in a restart analysis, a linear perturbation analysis, or other types of analyses. This enables you to control the contact status locally per contact pair. The key options that control contact interface behavior are: KEYOPT(12) of CONTA172, CONTA174, CONTA175, and CONTA177; and KEYOPT(10) of CONTA178.

You can change KEYOPT(12) to any value. For example, you can change from standard contact to bonded contact or vice-versa. If an open gap exists at the end of the previous load step and the contact status is adjusted to sliding or sticking due to a bonded or no-separation contact behavior definition, the program considers it as near-field contact when executing **CNKMOD (p. 392)** in the subsequent load steps.

In a linear perturbation analysis, the **CNKMOD (p. 392)** command adjusts the contact status from the linear perturbation base analysis (at the point of restart) as described in the table below. It enables you to take points in the base analysis that are near contact (within the pinball region) and modify them to be treated as in-contact in the perturbation analysis (see the "1 - near-field" row with KEYOPT(12) values set to 4 or 5). You can also take points that are sliding in the base analysis and treat them as sticking in the perturbation analysis, irrespective of the MU value (see the "2 - sliding" row with KEYOPT(12) values set to 1,3, 5, or 6).

Table 104: Contact Status Adjusted in Linear Perturbation Analysis via CNKMOD

Contact Status from the Base Analysis	CNKMOD,*ITYPE*,12,*Value*	
	KEYOPT(12) Value	Adjusted Contact Status

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution at the Restart Point		
0 - far-field	any	0 - far-field
1 - near-field	0, 1, 2, 3, 6	1 - near-field
	4	1 - near-field (if outside of the adjusted pinball region)
		2 - sliding (if inside of the adjusted pinball region)
	5	1 - near-field (if outside of the adjusted pinball region)
		3 - sticking (if inside of the adjusted pinball region)
2 - sliding	0, 2, 4	2 - sliding
	1, 3, 5, 6	3 - sticking
3 - sticking	any	3 - sticking

If an open gap exists at the end of the previous load step and the contact status is adjusted as sliding or sticking due to a bonded or no-separation contact behavior definition, the program considers it a near-field contact when executing **CNKMOD (p. 392)** in the subsequent load steps.

In the linear perturbation analysis procedure, contact status can also be controlled or modified via the **PERTURB (p. 1378)** command. The contact status always follows local controls defined by the **CNKMOD (p. 392)** command first, and is then adjusted by the global sticking or bonded setting (*ContKey* = STICKING or BONDED) on the **PERTURB (p. 1378)** command.

Modifying KEYOPT(3)

Another use for the **CNKMOD (p. 392)** command is to change the units of normal contact stiffness (contact element real constant FKN) in a linear perturbation modal analysis that is used to model brake squeal. For contact elements CONTA172 and CONTA174, KEYOPT(3) controls the units of normal contact stiffness. You can issue the command **CNKMOD (p. 392)**,*ITYPE*,3,1 during the first phase of the linear perturbation analysis in order to change the units of normal contact stiffness from FORCE/LENGTH^3 (in the base analysis) to FORCE/LENGTH. Note that KEYOPT(3) = 1 is valid only when a penalty-based algorithm is used (KEYOPT(2) = 0 or 1) and the absolute normal contact stiffness value is explicitly specified (that is, a negative value input for real constant FKN).

Modifying KEYOPT(15)

KEYOPT(15) controls the effect of contact stabilization damping for contact elements CONTA172, CONTA174, CONTA175, and CONTA177. You can use **CNKMOD (p. 392)** to activate or deactivate the contact

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

stabilization damping between load steps. For example, the command **CNKMOD (p. 392)**,*ITYPE*,15,1 deactivates the contact stabilization damping.

Menu Paths

This command cannot be accessed from a menu.

CNTR,*Option*,*Key*

Redirects contact pair output quantities to a text file.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Output option:

OUT

Contact output control.

Key

Control key:

NO

Write contact information to the output file or to the screen (default).

YES

Write contact information to the Jobname.cnm file.

Command Default

Contact pair output quantities are written to the output file (Jobname.out) or to the screen, as specified by the **/OUTPUT (p. 1334)** command.

Notes

Issue the command **CNTR (p. 395)**,OUT,YES to redirect contact pair output quantities to the Jobname.cnm file.

To ensure that the contact information is written to Jobname.cnm, reissue **CNTR (p. 395)**,OUT,YES each time you reenter the solution processor (**/SOLU (p. 1821)**).

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CNVTOL, *Lab*, *VALUE*, *TOLER*, *NORM*, *MINREF*

Sets convergence values for nonlinear analyses.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid convergence labels.

Discipline	Displacement Convergence		Force Convergence	
	Label	Label Description	Label	Label Description
Structural	U	Displacements	F	Forces
	ROT	Rotations	M	Moments
	HDSP	Hydrostatic Pressure	DVOL	Volume
	JOINT[a]	Joint Element Constraint Check		
	COMP[b]	Volumetric Compatibility Check		
	ENGY	System Energy Error Norm	ENGY	Energy Error[c]
Nonlocal field	GFV	Gradient Field Degree of Freedom	GFRS	Gradient Field Residual
Thermal	TEMP	Temperature	HEAT	Heat Flow
Fluid	PRES	Pressures	FLOW	Fluid Flow
	V	Velocity	VF	Fluid Force
Electric	VOLT	Voltage	AMPS or CHRG	Current Flow or Charge
	EMF	Electromotive Force	CURT	Current Flow
Magnetic	MAG	Scalar Potential	FLUX	Scalar Flux
	A	Vector Potential	CSG	Current Segments
Diffusion	CONC	Concentration	RATE	Diffusion Flow Rate

[a] JOINT specifies a tolerance for the joint element constraint check and is applicable to MPC184 joint elements. It is a nonlinear element convergence status value, not a displacement convergence value. *TOLER* is the only other valid argument when *Lab* = JOINT.

[b] COMP specifies a tolerance for the volumetric compatibility check in current-technology plane and solid elements (18x elements). It is a nonlinear element convergence status value, not a displacement convergence value. *TOLER* is the only other valid argument when *Lab* = COMP.

[c] ENGY specifies the convergence criterion for structural analysis based on the system of total energy accuracy. It is a combination of force and displacement criterion. The default energy error criterion is generally more relaxed than the default force criterion. Use ENGY to achieve a moderately accurate solution with a smaller number of iterations compared to the force convergence criterion.

VALUE

Typical reference value for the specified convergence label (*Lab*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VALUE defaults to the maximum of a program calculated reference or *MINREF*. For degrees of freedom, the reference is based upon the selected *NORM* and the current total degree-of-freedom value. For forcing quantities, the reference is based upon the selected *NORM* and the applied loads.

If *VALUE* is negative, the convergence criterion based on the specified label is removed, including the default convergence criterion value. The convergence criterion for all other labels remain as they were (either a default value or a previously specified value).

TOLER

Tolerance value used for the specified *Lab* convergence label. Default values are described below.

- If **CNVTOL (p. 396)** is issued with a *Lab* value specified but no *TOLER* value, the default tolerance values are:
  - 0.05 (5%) for displacement (U).
  - 1.0E-7 for the joint element constraint check (JOINT). This value rarely needs to be changed. A loose tolerance value may lead to inaccurate or incorrect solutions. When *Lab* = JOINT, *VALUE*, *NORM*, and *MINREF* are ignored.
  - 1.0E-3 for the volumetric compatibility check (COMP). When *Lab* = COMP, *VALUE*, *NORM*, and *MINREF* are ignored.
  - 0.05 for energy error (ENGY).
  - For all other *Lab* labels, the default tolerance value is 0.005 (0.5%).
- If **CNVTOL (p. 396)** is not issued, the *TOLER* defaults are as follows:
  - 0.005 (0.5%) for force (F) and moment (M)
  - 1.0E-4 (0.01%) for volume (DVOL)
  - 0.05 (5%) for displacement (U)
  - 0.05 (5%) for hydrostatic pressure (HDSP)
  - 1.0 for temperature (TEMP) when the iterative QUASI solver is used (**THOPT (p. 1997)**,QUASI,,,,,,1)
  - 1.0E-3 (0.1%) for acoustics pressure (PRES) and 1.E-3 (0.1%) for acoustic velocity potential using the degree of freedom PRES.

If you choose to specify a *TOLER* value, it must be greater than zero and less than 1. This is true for all *Lab* labels.

The program may adjust the force convergence tolerance if you do not explicitly set a value via **CNVTOL (p. 396)**. See Notes (p. 399) for details.

NORM

Specifies norm selection:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Infinite norm (check each degree of freedom separately) (default for *Lab* = U and for *Lab* = TEMP when the iterative QUASI solver is used (**THOPT (p. 1997)**,QUASI,,,,,,1).

The infinite norm is also used for the energy error criterion (ENGY) and is the only option available for ENGY.

1

L1 norm (check absolute value sum).

2

L2 norm (check SRSS value) (default, except for *Lab* = U).

3

Infinite norm (check each degree of freedom separately). The reference is calculated using the infinite norm of the displacement increment of the substep. Valid only for *Lab* = U.

MINREF

The minimum value allowed for the program-calculated reference value. If negative, no minimum is enforced. Used only if *VALUE* is blank. Default values are as follows:

- maximum of 0.01 or internally calculated minimum reference value for force (F), moment (M)
- a small factor times the average element length of the model for displacement (U) convergence
- 0.01 for volume (DVOL) convergence
- 1.0E-4 for gradient field residual (GFRS)
- 1.0E-6 for heat flow (HEAT)
- 1.0E-6 for diffusion flow (RATE)
- 1.0E-12 for charge (CHRG)
- 1.0E-6 for hydrostatic pressure (HDSP)
- 1.0 for temperature (TEMP) when the iterative QUASI solver is used (**THOPT (p. 1997)**,QUASI,,,,,,1)
- 1.0 for energy error convergence (ENGY)
- 1.0E-5 for acoustics pressure (PRES)
- 0.0 otherwise

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

For static or transient analysis, check the out-of-balance load for any active degree of freedom using the default *VALUE*, *TOLER*, *NORM*, and *MINREF*. Also check the translational displacement convergence in most cases. For harmonic magnetic analysis, check the out-of-balance of the degrees of freedom. The energy criterion convergence check is off by default.

Notes

This command is usually not needed because the default convergence criteria are sufficient for most nonlinear analyses. In rare cases, you may need to use this command to diagnose convergence difficulties.

Values may be set for the degrees of freedom and/or the out-of-balance load for the corresponding forcing quantities.

Issuing **CNVTOL (p. 396)** to set a convergence criterion for a specific convergence label (*Lab*) does not affect the convergence criterion for any other label. All other convergence criteria will remain at their default setting or at the value set by a previous **CNVTOL (p. 396)** command.

If **CNVTOL (p. 396)** is not issued for any force convergence label (F, M, DVOL, and so on as listed under the *Lab* argument), the default convergence tolerance for a particular force label is increased dynamically during the Newton-Raphson iterations in the range of 1 to 1.66 times the default value. For example, the F label default tolerance is 0.005. Therefore, the maximum convergence tolerance with the adjustment is 0.0083. This adjustment is not activated until the 8th or higher Newton-Raphson iteration. If you do not want the program to adjust the tolerance, issue **CNVTOL (p. 396)** to specify the convergence tolerance for the appropriate force label.

When using the Mechanical APDL graphical user interface (GUI), if a "Delete" operation in a Nonlinear Convergence Criteria dialog box writes this command to a log file (`Jobname.log` or `Jobname.lgw`), you will observe that *Lab* is blank, *VALUE* = -1, and *TOLER* is an integer number. In this case, the GUI has assigned a value of *TOLER* that corresponds to the location of a chosen convergence label in the dialog box's list. It is *not* intended that you type in such a location value for *TOLER* in an interactive session. However, a file that contains a GUI-generated **CNVTOL (p. 396)** command of this form can be used for batch input or with the **/INPUT (p. 948)** command.

Convergence norms specified with **CNVTOL (p. 396)** may be graphically tracked while the solution is in process using the Graphical Solution Tracking (GST) feature. Issue **/GST (p. 880)** to enable or disable GST. By default, GST is ON for interactive sessions and OFF for batch runs.

The energy convergence check (ENGY) is not available when the arc-length method (**ARCLEN (p. 175)**,ON) is used.

For more information on convergence calculations in a nonlinear analysis, see Convergence in the *Theory Reference*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Convergence Crit

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Harmonic

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Static

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Transient

Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Solution>Load Step Opts>Nonlinear>Convergence Crit

Main Menu>Solution>Load Step Opts>Nonlinear>Harmonic

Main Menu>Solution>Load Step Opts>Nonlinear>Static

Main Menu>Solution>Load Step Opts>Nonlinear>Transient

/COLOR, *Lab*, *Clab*, *N1*, *N2*, *NINC*

Specifies the color mapping for various items.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Apply color to the items specified by the following labels:

AXES

Determines the color (specified in next argument, *Clab*) that the axes of a graph will be plotted in.

AXNUM

Determines the color (specified in next argument, *Clab*) that the numbering on the axes of a graph will be plotted in.

NUM

Discretely numbered items (such as element types, element materials, etc., as shown on the **/PNUM (p. 1457)** command). Also specify number (1 to 11) in the *N1* field. For example, **/COLOR (p. 400)**,NUM,RED,3 will assign the color red to all items having the discrete number 3 (material displays would show elements having material 3 as red).

OUTL

Outline of elements, areas, and volumes. Ex: **/COLOR (p. 400)**,OUTL,BLUE.

ELEM

Elements. Use *N1*, *N2*, *NINC* fields for element numbers.

LINE

Solid model lines. Use *N1*, *N2*, *NINC* fields for line numbers.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AREA

Solid model areas. Use *N1*, *N2*, *NINC* fields for area numbers.

VOLU

Solid model volumes. Use *N1*, *N2*, *NINC* fields for volume numbers.

ISURF

Isosurfaces (surfaces of constant stress, etc.). This option is particularly useful when capturing frames for animating a single isosurface value.

WBAK

Window background. Use *N1*, *N2*, *NINC* fields for window numbers. The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

b.c.label

Boundary condition label. Enter U, ROT, TEMP, PRES, V, VOLT, MAG, A, EMF, CURR, F, M, HEAT, FLOW, VF, AMPS, FLUX, CSG, CURT, MAST, CP, CE, NFOR, NMOM, RFOR, RMOM, PATH. See the **/PBC (p. 1356)** command for boundary condition label definitions.

GRBAK

Graph background.

GRID

Graph grid lines.

AXLAB

Graph X and Y axis labels.

CURVE

Graph curves (identify curve numbers (1-10) in *N1*, *N2*, *NINC* fields).

CM

Component group. Use *N1* field for component name, ignore *N2* and *NINC*.

CNTR

Mechanical APDL contour stress colors. The maximum number of contours available is 128. The number of colors that can be specified interactively (GUI) is 9. (**/CONTOUR (p. 415)**, , 9). Any other setting will yield inconsistent results.

SMAX

Specifies that all stress values above the maximum value entered in **/CONTOUR (p. 415)** will be displayed in the color designated in the *Clab* field. Defaults to dark grey.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SMIN

Specifies that all stress values below the minimum value entered in **/CONTOUR (p. 415)** will be displayed in the color designated in the *Clab* field. Defaults to dark grey.

PBAK

Activates background shading options (see command syntax at end of argument descriptions below). The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

Clab

Valid color labels are:

BLAC (0)

Black

MRED (1)

Magenta-Red

MAGE (2)

Magenta

BMAG (3)

Blue-Magenta

BLUE (4)

Blue

CBLU (5)

Cyan-Blue

CYAN (6)

Cyan

GCYA ((7)

Green-Cyan

GREE (8)

Green

YGRE (9)

Yellow-Green

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

YELL (10)

Yellow

ORAN (11)

Orange

RED (12)

Red

DGRA (13)

Dark Gray

LGRA (14)

Light Gray

WHIT (15)

White

N1, N2, NINC

Apply color to *Lab* items numbered *N1* to *N2* (defaults to *N1*) in steps of *NINC* (defaults to 1). If *N1* is blank, apply color to entire selected range. If *Lab* is CM, use component name for *N1* and ignore *N2* and *NINC*. If *N1* = P, graphical picking of elements, lines, areas and volumes is enabled; your can assign colors to the entities via the picker. When picking is enabled, the *Lab* and *Clab* fields are ignored.

If *Lab* = PBAK, the command format is **/COLOR (p. 400)**, PBAK,*Key_On_Off*, *KEY_TYPE*, *KEY_INDEX*.

The options that you select using *Lab* = PBAK will supersede those applied using *Lab* = WBAK.

Key_On_Off

Turns the background colors on and off. Acceptable values are ON (1) and OFF (0).

KEY_TYPE

Determines the type of background. Acceptable values are 0 (smooth shading left to right), 1 (smooth shading top to bottom), 2 (smooth shading right to left), 3 (smooth shading bottom to top), and -1 (textured image background)

KEY_INDEX

An integer value that corresponds to a background color or texture. If *Key_Type* is -1, the background will correspond to values specified in the **/TXTRE (p. 2029)** command. If *Key_Type* is any other acceptable value, the background will correspond to the color values listed above under *Clab*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Use the default color mapping.

Notes

Issue **/COLOR (p. 400)**,STAT to display the current color mapping. Issue **/COLOR (p. 400)**,DEFA to reset the default color mapping.

Note:

Color labels may also be reassigned any "color" with the **/CMAP (p. 359)** command.

This command is valid anywhere.

Menu Paths

Utility Menu>PlotCtrls>Style>Colors>BC Colors
Utility Menu>PlotCtrls>Style>Colors>Component Colors
Utility Menu>PlotCtrls>Style>Colors>Entity Colors
Utility Menu>PlotCtrls>Style>Colors>Graph Colors
Utility Menu>PlotCtrls>Style>Colors>Numbered Item Colors
Utility Menu>PlotCtrls>Style>Colors>Window Colors

/COM, *Comment*

Places a comment in the output.

SESSION (p. 11): List Controls (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Comment

Comment string, up to 75 characters.

Notes

The output from this command consists of the comment string. This command is similar to **C*** (p. 283)** except that the comment produced by **C*** (p. 283)** is more easily identified in the output. Parameter substitution within the comment occurs for every valid expression delimited by percent (%) signs. Enclosing such an expression in single quotes prevents parameter substitution.

Another way to include a comment is to precede it with a ! character (on the same line). The ! may be placed anywhere on the line, and any input following it is ignored as a comment. No output is produced by such a comment, but the comment line is included on the log file. This is a convenient way to annotate the log file.

This command is valid anywhere.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

---

*COMP,*Matrix*,*Algorithm*,*THRESHOLD*,*Val1*,*Val2*

Compresses a matrix using a specified algorithm.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Matrix

Name of the matrix to compress.

Algorithm

Algorithm or method to use:

SVD --

Singular value decomposition algorithm (default).

MGS --

Modified Gram-Schmidt algorithm.

SPARSE --

Compress a sparse matrix based on the threshold value.

THRESHOLD

Numerical threshold value used to manage the compression. The default value depends on the method of compression: 1E-7 for SVD; 1E-14 for MGS; 1E-16 for SPARSE.

Val1 and *Val2* are additional input used only for the SVD algorithm:

Val1

Name of the vector used to store the Σ values (see "Notes" (p. 406) below). This argument is optional.

Val2

Name of the dense matrix used to store the $\mathbf{V}^*$ output matrix (see "Notes" (p. 406) below). This argument is optional.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The SVD and MGS algorithms are only applicable to dense matrices that were created using the ***DMAT (p. 551)** command. Columns that are linearly dependent on others are removed, leaving the independent or basis vectors. The matrix is resized according to the new size determined by the algorithm.

For the SVD algorithm, the singular value decomposition of an input matrix $\mathbf{M}$ is a factorization of the form:

$$\mathbf{M}=\mathbf{U}\Sigma\mathbf{V}^{*}$$

Here, the $\mathbf{M}$ matrix is replaced by the $\mathbf{U}$ matrix, according to the specified threshold.

The SPARSE compression method is only applicable to sparse matrices that were created using the ***SMAT (p. 1801)** command. All terms that have an absolute value below the specified threshold, relative to the maximum value in the matrix, are removed from the original matrix. For example, given a sparse matrix having 100 as the largest term and *THRESHOLD* = 0.5, all terms having an absolute value below 0.5*100 = 50 are removed.

Menu Paths

This command cannot be accessed from a menu.

COMBINE, *FileType*, *NUM*

Combines distributed memory parallel (DMP) files.

AUX2 (p. 61): Binary File Manipulation (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FileType

Type of solution file to combine. There is no default; if (blank), the command is ignored.

RST

Structural results file (`.rst`)

RTH

Thermal results file (`.rth`)

RMG

Magnetics results file (`.rmg`)

RSTP

Linear perturbation results file (`.rstp`)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMAT

Element matrix file (`.emat`).

ESAV

Element saved data file (`.esav`)

MODE

Modal results file (`.mode`)

MLV

Modal load vector file (`.mlv`)

IST

Initial state file (`.ist`)

FULL

Full matrix file (`.full`)

Reduced complex displacement file (`.`)

RDSP

Reduced displacement file (`.rdsp`)

RNNN

Multiframe restart files (`.r`*nnn*)

NUM

Number of `.r`*nnn* files to combine:

ALL

Combine all `.r`*nnn* files (default).

N

Combine only `.r`*n* files, where N is an integer from 1 to 999.

Notes

The **COMBINE (p. 406)** command is used within the AUX2 auxiliary processor to combine local solution files from a distributed memory parallel solution into a single, global file. Before using this command, you must enter the AUX2 processor by issuing the **/AUX2 (p. 208)** command.

In a distributed-memory parallel (DMP) solution, you can use the **DMPOPTION (p. 559)** command to bypass the file combination step, causing all individual local files to be kept on the local disks in the current working directory. Later on, you can start a new distributed memory parallel solution and use

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the **COMBINE (p. 406)** command to combine local files into a global file for a downstream solution or another operation (such as postprocessing with **/POST1 (p. 1461)**). For example, the command **COMBINE (p. 406)**,RST will combine local results files (`Jobname`*n*`.rst`) into a global results file (`Jobname.rst`).

The **COMBINE (p. 406)** command cannot be used to combine local files generated during a distributed memory parallel solution that used the frequency or cyclic harmonic index domain decomposition method (**DDOPTION (p. 508)**,FREQ or **DDOPTION (p. 508)**,CYCHI).

If **COMBINE (p. 406)**,RNNN is specified, all of the multiframe restart files named `Jobname.r001` to `Jobname.r999` will automatically be combined. To combine only one set of `.r`*nnn* restart files, place only that set of restart files in your current working directory, or use the *NUM* argument to specify which set of `.r`*nnn* files to combine.

When the **COMBINE (p. 406)** command is used in a subsequent distributed memory parallel (DMP) session, the number of processors must be the same as in the DMP solution that generated the files.

When running on a cluster, the local solution files must be available in the working directory on each node in the subsequent session. As an example, consider the following command line used to generate local solution files:

```
ansys241 -dis -machines machine1:4:machine2:1:machine3:2 -i input -o output
```

Different machines can be used in the subsequent session to combine these files. However, the total number of cores must remain unchanged (seven in the above case), and the local files must be copied to the working directory (or directories) on each of the machines used in the subsequent session.

Menu Paths

This command cannot be accessed from a menu.

COMPRESS

Deletes all specified sets.

AUX3 (p. 61): Results Files (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Issue this command to delete all sets specified with the **DELETE (p. 513)** command.

The **COMPRESS (p. 408)** command is valid only in the results file editing processor (auxiliary processor AUX3), and, like the other AUX3 commands, it only affects the data steps index (DSI), time (TIM), loadstep, substep and cumulative step iteration (LSP) records in the results file.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CON4, *XCENTER*, *YCENTER*, *RAD1*, *RAD2*, *DEPTH*

Creates a conical volume anywhere on the working plane.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCENTER, YCENTER

Working plane X and Y coordinates of the center axis of the cone.

RAD1, RAD2

Radii of the faces of the cone. *RAD1* defines the bottom face and will be located on the working plane. *RAD2* defines the top face and is parallel to the working plane. A value of zero or blank for either *RAD1* or *RAD2* defines a degenerate face at the center axis (that is, the vertex of the cone). The same value for both *RAD1* and *RAD2* defines a cylinder instead of a cone.

DEPTH

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cone. *DEPTH* cannot be zero (see "Notes" (p. 409) below).

Notes

Defines a solid conical volume with either the vertex or a face anywhere on the working plane. The cone must have a spatial volume greater than zero. (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The face or faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°). See the **CONE (p. 409)** command for an alternate way to create cones.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Primitives>Cone

Main Menu>Preprocessor>Modeling>Create>Volumes>Cone>By Picking

CONE, *RBOT*, *RTOP*, *Z1*, *Z2*, *THETA1*, *THETA2*

Creates a conical volume centered about the working plane origin.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RBOT, RTOP

Radii of the bottom and top faces of the cone. A value of zero or blank for either *RBOT* or *RTOP* defines a degenerate face at the center axis (that is, the vertex of the cone). The same value for both *RBOT* and *RTOP* defines a cylinder instead of a cone.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z1, Z2

Working plane Z coordinates of the cone. The smaller value is always associated with the bottom face.

THETA1, THETA2

Starting and ending angles (either order) of the cone. Used for creating a conical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°. See the *Modeling and Meshing Guide* for an illustration.

Notes

Defines a solid conical volume centered about the working plane origin. The non-degenerate face (top or bottom) is parallel to the working plane but not necessarily coplanar with (that is, "on") the working plane. The cone must have a spatial volume greater than zero. (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) For a cone of 360°, top and bottom faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°). See the **CON4 (p. 409)** command for an alternate way to create cones.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Cone>By Dimensions

---

/CONFIG, *Lab*, *VAL*

Assigns values to Mechanical APDL configuration parameters.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Configuration parameter to be changed:

NORSTGM

Option to write or not write geometry data to the results file:

When *VAL* = 0, write geometry data (default).

When *VAL* = 1, does not write geometry data.

Useful when complex analyses are likely to create abnormally large files.

NBUF

The number of buffers (*VAL* = 1 to 32) per file in the solver. Default: *VAL* = 4.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LOCFL

File open and close actions:

When *VAL* = 0, global (default).

When *VAL* = 1, local.

Applies to `File.erot`, `File.esav`, and `File.emat`.

Typically used for large problems where locally closed files may be deleted earlier in the run via **/FDELE (p. 744)**.

SZBIO

Record size (*VAL* = 1024 to 4194304) of binary files (in integer words).

Default: *VAL* = 16384 (system-dependent).

FSPLIT

Defines split points for binary files. *VAL* is the file split point in megawords.

Default: *VAL* = Maximum file size for the system.

MXND

Maximum number of nodes.

Default: *VAL* = 100 at first encounter.

Dynamically expanded by doubling, even at first encounter, when the maximum is exceeded.

MXEL

Maximum number of elements. Default and expansion as for MXND.

MXKP

Maximum number of keypoints. Default and expansion as for MXND.

MXLS

Maximum number of lines. Default and expansion as for MXND.

MXAR

Maximum number of areas. Default and expansion as for MXND.

MXVL

Maximum number of volumes. Default and expansion as for MXND.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MXRL

Maximum number of sets of real constants (element attributes). Default and expansion as for MXND.

MXCP

Maximum number of sets of coupled degrees of freedom. Default and expansion as for MXND.

MXCE

Maximum number of constraint equations. Default and expansion as for MXND.

NOELDB

Option to write or not write results into the database after a solution.

When *VAL* = 0 (default), writes results into the database.

When *VAL* = 1, does not write results into the database.

NUMLV

Maximum number of load vectors written on `Jobname.mode` file when *MSUPkey* = YES on the **MXPAND (p. 1203)** command.

Default: *VAL* = 1000 at first encounter.

When the maximum is exceeded, the value is not expanded.

The NUMLV option is not supported for fast load vector generation (*FastLV* = ON on the **MODCONT (p. 1132)** command).

NUMSUBLV

Maximum number of load vectors written on `Jobname.sub` file in substructure/CMS generation pass.

Default: *VAL* = 31 at first encounter.

When the maximum is exceeded, the value is not expanded.

GRW_NBUF

Option to automatically grow the number of file buffers for most binary files (`.esav`, `.emat`, `.full`, and so on), with the exception of the results file and files written by the sparse and PCG equation solvers (for example, `.DSP`*xxxx* and `.pc`*n*).

When *VAL* = -1, the number of file buffers does not grow automatically for any file.

When *VAL* = 0 (default), the number of file buffers may or may not grow automatically. The logic is program-controlled.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When *VAL* = 1, the number of file buffers automatically grows for most binary files to reduce the amount of I/O. This option may require a significantly greater amount of memory than the default behavior (*VAL* = 0).

MEBA_LIC

Option to control automatic checkout of a Mechanical batch license during solution when the capability is not enabled, useful for PrepPost sessions:

When *VAL* = 0 (default), check out a license automatically when needed.

When *VAL* = 1, bypass automatic license checkout.

STAT

Displays current values set by the **/CONFIG (p. 410)** command.

VAL

Value (an integer number) assigned to the specified configuration parameter.

Notes

All configuration parameters have initial defaults, which in most cases do not need to be changed. Where a specially configured version of the Mechanical APDL program is desired, the parameters can be changed with this command.

Issue **/CONFIG (p. 410)**,STAT to display current values.

Define changes before the parameter is required.

These changes (and others) may also be incorporated into the `config.ans` file, read in upon execution of the program. (See The Configuration File in the *Basic Analysis Guide*.) If the same configuration parameter appears in both the configuration file and this command, this command overrides.

Distributed-memory parallel DMP solutions use the default FSPLIT value and force NOELDB = 1 for all results files. You cannot change the FSPLIT and NOELDB options for a DMP solution.

Menu Paths

Utility Menu>List>Status>Configuration

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONJUG, *IR*, *IA*, - - , - - , *Name*, - - , - - , *FACTA*

Forms the complex conjugate of a variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

--,--

Unused fields.

Name

Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--,--

Unused fields.

FACTA

Scaling factor (positive or negative) applied to variable (default to 1.0).

Notes

Used only with harmonic analyses (**ANTYPE (p. 162)**,HARMIC).

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Complex Conjugat

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/CONTOUR, *WN*, *NCONT*, *VMIN*, *VINC*, *VMAX*

Specifies the uniform contour values on stress displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

NCONT

Number of contour values. *NCONT* defaults to 9 for X11 or WIN32 and to 128 for X11c or WIN32C. The default graphics window display for 3D devices is a smooth continuous shading effect that spans the maximum of 128 contours available. Use the **/DV3D (p. 584)** command to create defined banding for your contour values (values of 9 and 128 are displayed in smooth shading only). The legend, however, will display only nine color boxes, which span the full range of colors displayed in the graphics window.

VMIN

Minimum contour value. If *VMIN* = AUTO, automatically calculate contour values based upon *NCONT* uniformly spaced values over the min-max extreme range. Or, if *VMIN* = USER, set contour values to those of the last display (useful when last display automatically calculated contours).

VINC

Value increment (positive) between contour values. Defaults to (*VMAX*-*VMIN*)/*NCONT*.

VMAX

Maximum contour value. Ignored if both *VMIN* and *VINC* are specified.

Command Default

Nine contour values uniformly spaced between the extreme values, or no contours if the ratio of range to minimum value (or range to maximum if minimum = 0) is less than 0.001.

Notes

Values represent contour lines in vector mode, and the algebraic maximum of contour bands in raster mode.

Regardless of how many contours (*NCONT*) are specified, the *actual* number of contours appearing on your display depends also on the device name, whether the display is directed to the screen or to a file, the display mode (vector or raster), and the number of color planes. (All of those items are controlled via **/SHOW (p. 1785)**.) In any case, regardless of whether they are smoothed or banded, only 128 contours can be displayed.

For more information about changing the number of contours, see Creating Geometric Results Displays.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the current Mechanical APDL graphics are *not* displayed as Multi-Plots:

If the current device is a 3D device (**/SHOW (p. 1785)**,3D), the model contours in all active windows are the same, even if separate **/CONTOUR (p. 415)** commands are issued for each active window.

Mechanical APDL maintains a single data structure (segment) containing one set of contours. The program displays the same segment in all windows. The view settings of each window constitute the only differences in the contour plots in the active windows.

This command is valid in any processor.

For alternate specifications, see **/CVAL (p. 450)**.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Uniform Contours

/COPY,*Fname1*,*Ext1*,--,*Fname2*,*Ext2*,--,*DistKey*

Copies a file.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname1

File name to be copied and its directory path (248 characters maximum for both file name and directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

The file name defaults to the current Jobname.

Ext1

Filename extension (eight-character maximum).

--

Unused field.

Fname2

File name to be created and its directory path (248 characters maximum for both file name and directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

Fname2 defaults to *Fname1*.

Ext2

Filename extension (eight-character maximum).

Ext2 defaults to *Ext1*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--

Unused field.

DistKey

Key that specifies which copy operation is performed on all processes in distributed-memory parallel mode :

0 (OFF or NO)

The program performs the copy operation only on the master process (default).

1 (ON or YES)

The program performs the copy operation locally on each process.

2 or BOTH

The program performs the copy operation for *Fname*.*Ext* on the master process and for *FnameN*.*Ext* on all processes.

Notes

The original file is untouched. Ex: **/COPY (p. 416)**,A,,,B copies file A to B in the same directory. **/COPY (p. 416)**,A,DAT,,,INP copies the file A.DAT to A.INP. See the *Operations Guide* for details. Mechanical APDL binary and ASCII files can be copied.

In distributed-memory parallel (DMP) mode, only the master process will copy *Fname1*.*Ext1* to *Fname2*.*Ext2* by default. However, when *DistKey* is set to 1 (or ON, or YES) or 2 (or BOTH), the command is executed by all processes. In this case, *Fname1* and *Fname2* will automatically have the process rank appended to them. This means *Fname1N*.*Ext1* will be copied to *Fname2N*.*Ext2* by all processes, where *N* is the DMP process rank. For more information see Differences in General Behavior in the *Parallel Processing Guide*.

Menu Paths

Utility Menu>File>File Operations>Copy

CORIOLIS,*Option*,*--*,*--*,*RefFrame*,*RotDamp*,*RotMass*

Applies the Coriolis effect to a rotating structure.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Flag to activate or deactivate the Coriolis effect:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 (ON or YES)

Activate. This value is the default.

0 (OFF or NO)

Deactivate.

--,---

Unused fields.

RefFrame

Flag to activate or deactivate a stationary reference frame.

1 (ON or YES)

Activate.

0 (OFF or NO)

Deactivate. This value is the default.

RotDamp

Flag to activate or deactivate rotating damping effect.

1 (ON or YES)

Activate.

0 (OFF or NO)

Deactivate. This value is the default.

RotMass

Flag to activate or deactivate rotor mass summary printout (only supported for *RefFrame* = ON).

1 (ON or YES)

Activate.

0 (OFF or NO)

Deactivate. This value is the default.

Notes

The **CORIOLIS (p. 417)** command is used for linear analyses in either a rotating or a stationary reference frame, and performs differently according to the designated *RefFrame* value. The **CORIOLIS (p. 417)** command must be specified during the first step of the analysis. The rotational velocity must be defined using **OMEGA (p. 1324)** (when the whole model is rotating) or **CMOMEGA (p. 368)** (component based rotation). Specific restrictions and elements apply to each case, as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotating Reference Frame (*RefFrame* = OFF)

The command applies the Coriolis effect in the following structural element types: MASS21, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL281, PIPE288 and PIPE289. It also applies this effect in the following coupled-field elements when structural degrees of freedom are present: PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227.

The rotating damping effect (*RotDamp* = ON) is only supported by the COMBI214 element when stationary.

In a rotating reference frame, the Coriolis and spin-softening effects, as well as the centrifugal forces, contribute to the dynamics and are applied by default.

Stationary Reference Frame (*RefFrame* = ON)

The command activates the gyroscopic damping matrix in the following structural elements: MASS21, BEAM188, SHELL181, BEAM189, SOLID185, SOLID186, SOLID187, SOLSH190, SOLID272, SOLID273, SHELL281, PIPE288, PIPE289, and MATRIX50.

The rotating structure is assumed to be axisymmetric about the axis of rotation.

The rotating damping effect (*RotDamp* = ON) is supported by the elements listed above that generate a gyroscopic damping matrix. It is also supported by some specific elements (see Adding Damping in the *Rotordynamic Analysis Guide* for a complete list).

The rotor mass summary printout (*RotMass* = ON) is only supported for some of the elements that generate a gyroscopic damping matrix: MASS21, BEAM188, BEAM189, PIPE288, and PIPE289. The EMAT file is required (**EMATWRITE (p. 623)**,YES).

To include Coriolis effects in a linear perturbation (prestressed) analysis, follow the procedure detailed in Considerations for Rotating Structures.

In a nonlinear transient analysis in which the model is actually spinning (**ANTYPE (p. 162)**,TRANS and **NLGEOM (p. 1231)**,ON) the **CORIOLIS (p. 417)** command must not be used as any spinning motion applied through either the **IC (p. 921)** or **D (p. 483)** commands automatically includes nonlinear inertia terms such as the Coriolis effect.

To take into account variable bearings (COMBI214 elements with tabular user-defined characteristics), you must activate the Coriolis effect in a stationary reference frame. The gyroscopic effect coming from COMBI214 mass characteristics is not supported.

For more information about using the **CORIOLIS (p. 417)** command, see Rotating Structure Analysis in the *Advanced Analysis Guide* and also Overview of the Rotordynamic Analysis Process in the *Rotordynamic Analysis Guide*. For details about the Coriolis and gyroscopic effect element formulations, see the *Mechanical APDL Theory Reference*.

Elements with layered section properties do not support Coriolis effects (rotating and stationary reference frames).

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>Coriolis

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>Coriolis

---

COUPLE

Specifies "Node coupling" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Coupled Sets

COVAL, *TBLN01, TBLN02, SV1, SV2, SV3, SV4, SV5, SV6, SV7*

Defines PSD cospectral values.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLN01

First input PSD table number associated with this spectrum.

TBLN02

Second input PSD table number associated with this spectrum.

SV1, SV2, SV3, . . . , SV7

PSD cospectral values corresponding to the frequency points (**PSDFRQ (p. 1528)**).

Notes

Defines PSD cospectral values to be associated with the previously defined frequency points. Two table references are required since values are off-diagonal terms. Unlike autospectra (**PSDVAL (p. 1534)**), the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cospectra can be positive or negative. The cospectral curve segment where there is a sign change is interpolated linearly (the rest of the curve segments use log-log interpolation). For better accuracy, choose as small a curve segment as possible wherever a sign change occurs.

Repeat **COVAL (p. 420)** command using the same table numbers for additional points. This command is valid for **SPOPT (p. 1837)**,PSD only.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Cospectral

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Cospectral

---

CP, *NSET*, *Lab*, *NODE1*, *NODE2*, *NODE3*, *NODE4*, *NODE5*, *NODE6*, *NODE7*, *NODE8*, *NODE9*, *NODE10*, *NODE11*, *NODE12*, *NODE13*, *NODE14*, *NODE15*, *NODE16*, *NODE17*

Defines (or modifies) a set of coupled degrees of freedom.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET

Set reference number:

n

Arbitrary set number.

HIGH

The highest defined coupled set number will be used (default, unless *Lab* = ALL). This option is useful when adding nodes to an existing set.

NEXT

The highest defined coupled set number plus one will be used (default if *Lab* = ALL). This option automatically numbers coupled sets so that existing sets are not modified.

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). Defaults to label previously defined with *NSET* if set *NSET* already exists. A different label redefines the previous label associated with *NSET*. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations) (in radians); HDSP (hydrostatic pressure). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential); CURR (current). Diffusion label: CONC (concentration).

When *Lab* = ALL:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sets are generated for *each* active degree of freedom (that is, one set for the UX degree of freedom, another set for UY, etc.), and *NSET* is incremented automatically to prevent overwriting existing sets.
- Existing sets are *not* modified. *NSET* must be a new set number *n* or NEXT.
- The degree of freedom set is determined according to all element types defined and the **DOF (p. 568)** command, if used.
- Hydrostatic pressure (HDSP) is not included.

NODE1, NODE2, NODE3, . . . , NODE17

List of nodes to be included in set. Duplicate nodes are ignored. If a node number is input as negative, the node is deleted from the coupled set. The first node in the list is the primary (retained) node, and the remaining nodes represent the removed degrees of freedom.

If *NODE1* = ALL, *NODE2* through *NODE17* are ignored and all selected nodes (**NSEL (p. 1266)**) are included in the set, and the node with the lowest node number becomes the primary node.

If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

A component name can be substituted for *NODE1*. The component consists of the node group to be coupled. The node with the lowest node number becomes the primary node among the node group. To display the generated and coupled node sets, issue the **CPLIST (p. 427)** command.

Notes

Do not include the same degree of freedom in more than one coupled set. Repeat **CP (p. 421)** command for additional nodes.

Coupling degrees of freedom into a set causes the results calculated for one member of the set to be the same for all members of the set. Coupling can be used to model various joint and hinge effects. A more general form of coupling can be done with constraint equations (**CE (p. 297)**). For structural analyses, a list of nodes is defined along with the nodal directions in which these nodes are to be coupled. As a result of this coupling, these nodes are forced to take the same displacement in the specified *nodal coordinate direction*. The amount of the displacement is unknown until the analysis is completed. A set of coupled nodes which are not coincident, or which are not along the line of the coupled displacement direction, may produce an applied moment which will not appear in the reaction forces. The actual degrees of freedom available for a particular node depends upon the degrees of freedom associated with element types (**ET (p. 686)**) at that node. For scalar field analysis, this command is used to couple nodal temperatures, pressures, voltages, etc.

A set of coupled nodes which are not coincident, or which are not along the line of the coupled displacement direction, produce an artificial moment constraint. If the structure rotates, a moment may be produced in the coupled set in the form of a force couple. This moment is in addition to the real reaction forces and may make it appear that moment equilibrium is not satisfied by just the applied forces and the reaction forces.

Additional sets of coupled nodes may be generated from a specified set. Degrees of freedom are coupled within a set but are not coupled between sets. No degree of freedom should appear in more than one coupled set. Such an appearance would indicate that at least two sets were in fact part of a single larger

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

set. The first degree of freedom of the coupled set is the "prime" degree of freedom. All other degrees of freedom in the coupled sets are eliminated from the solution matrices by their relationship to the prime degree of freedom. Forces applied to coupled nodes (in the coupled degree of freedom direction) will be summed and applied to the prime degree of freedom. Output forces are also summed at the prime degree of freedom. Degrees of freedom with specified constraints (**D (p. 483)**) should not be included in a coupled set (unless the degree of freedom is prime).

If master degrees of freedom are defined for coupled nodes, only the prime degree of freedom should be so defined. The use of coupled nodes reduces the set of coupled degrees of freedom to only one degree of freedom.

The removed degrees of freedom defined by the **CP (p. 421)** command cannot be included in any **CE (p. 297)** or **CERIG (p. 310)** command.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Couple DOFs

Main Menu>Preprocessor>Coupling / Ceqn>Cupl DOFs w/Indep

CPCYC, *Lab*, *TOLER*, *KCN*, *DX*, *DY*, *DZ*, *KNONROT*, *KMID*, *CEOPT*

Couples the two side faces of a cyclically symmetric model for loadings that are the same on every segment.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). If ALL, use all appropriate labels. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians). Thermal label: TEMP (temperature). Fluid label: PRES (pressure). Electric label: VOLT (voltage).

TOLER

Tolerance for coincidence (based on maximum coordinate difference in each global Cartesian direction for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident for coupling.

KCN

In coordinate system *KCN*, two nodes are coupled when the coordinates of the first node (on the low boundary) plus the increments DX, DY, and DZ match the coordinates of the second node (on the high boundary).

DX, DY, DZ

Node location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, D θ, DΦ for spherical or toroidal).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KNONROT

When *KNONROT* = 0, the nodes on coupled sets are rotated into coordinate system *KCN* (see **NROTAT (p. 1264)** command description). When *KNONROT* = 1, the nodes are not rotated, and you should make sure that coupled nodal DOF directions are correct.

KMID

When *KMID* = 1, the midside nodes of the element edges are added to the coupled sets of the end nodes of edges with specified nodal DOFs, if the end nodes of an edge are coupled. By default (*KMID* = 0), the midside nodes are not included in the coupled sets.

CEOPT

When *CEOPT* = 1, the coupled sets are converted to constraint equations. Use this option to improve performance in a distributed-memory parallel (DMP) solution. By default (*CEOPT* = 0), the coupled sets are not converted to constraint equations.

Notes

Cyclic coupling requires identical node and element patterns on the low and high sector boundaries. The **MSHCOPY (p. 1189)** operation allows convenient generation of identical node and element patterns. See Using CPCYC and MSHCOPY Commands in the *Modeling and Meshing Guide* for more information.

Although developed initially for use with cyclically symmetric models, use of the **CPCYC (p. 423)** command is not limited to cyclic symmetry analyses.

Example Usage

Using CPCYC and MSHCOPY Commands

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Offset Nodes

CPDELE, *NSET1*, *NSET2*, *NINC*, *Nsel*

Deletes coupled degree of freedom sets.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET1*, *NSET2*, *NINC

Delete coupled sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL, *NSET2* and *NINC* are ignored and all coupled sets are deleted.

Nsel

Additional node selection control:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANY

Delete coupled set if *any* of the selected nodes are in the set (default).

ALL

Delete coupled set only if *all* of the selected nodes are in the set.

Notes

See the **CP (p. 421)** command for a method to delete individual nodes from a set.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Del Coupled Sets

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

---

CPINTF, *Lab*, *TOLER*

Defines coupled degrees of freedom at an interface.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). If ALL, use all appropriate labels *except* HDSP. Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations, in radians), HDSP (hydrostatic pressure). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential); CURR (current). Diffusion label: CONC (concentration).

TOLER

Tolerance for coincidence (based on maximum coordinate difference in each global Cartesian direction for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident for coupling.

Notes

Defines coupled degrees of freedom between coincident nodes (within a tolerance). May be used, for example, to "button" together elements interfacing at a seam, where the seam consists of a series of node pairs. One coupled set is generated for each selected degree of freedom for each pair of coincident nodes. For more than two coincident nodes in a cluster, a coupled set is generated from the lowest numbered node to each of the other nodes in the cluster. Coupled sets are generated only within (and not between) clusters. If fewer than all nodes are to be checked for coincidence, use the **NSEL (p. 1266)** command to select nodes. Coupled set reference numbers are incremented by one from the highest previous set number. Use **CPLIST (p. 427)** to display the generated sets. Only nodes having the same nodal coordinate system orientations ("coincident" within a tolerance) are included. Use the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CEINTF (p. 305) command to connect nodes by constraint equations instead of by coupling. Use the **EINTF (p. 614)** command to connect nodes by line elements instead of by coupling.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Coincident Nodes

/CPLANE, *KEY*

Specifies the cutting plane for section and capped displays.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Specifies the cutting plane:

0

Cutting plane is normal to the viewing vector (**/VIEW (p. 2131)**) and passes through the focus point (**/FOCUS (p. 771)**) (default).

1

The working plane (**WPLANE (p. 2180)**) is the cutting plane.

Command Default

The cutting plane is normal to the viewing vector at the focus point.

Notes

Defines the cutting plane to be used for section and capped displays (**/TYPE (p. 2034)**,,(1, 5, or 7)).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPLGEN, *NSETF*, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*

Generates sets of coupled nodes from an existing set.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSETF

Generate sets from existing set *NSETF*.

Lab1*, *Lab2*, *Lab3*, . . . , *Lab5

Generate sets with these labels (see **CP (p. 421)** command for valid labels). Sets are numbered as the highest existing set number + 1.

Notes

Generates additional sets of coupled nodes (with different labels) from an existing set (**CP (p. 421)**, **CPNGEN (p. 430)**). The same node numbers are included in the generated sets. If all labels of nodes are to be coupled and the nodes are coincident, the **NUMMRG (p. 1295)** command should be used to automatically redefine the node number (for efficiency).

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same Nodes

CPLIST, *NSET1*, *NSET2*, *NINC*, *Nsel*

Lists the coupled degree of freedom sets.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET1*, *NSET2*, *NINC

List coupled sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL (default), *NSET2* and *NINC* are ignored and all coupled sets are listed.

Nsel

Node selection control:

ANY

List coupled set if *any* of the selected nodes are in the set (default).

ALL

List coupled set only if *all* of the selected nodes are in the set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Coupled Sets>All CP nodes selected

Utility Menu>List>Other>Coupled Sets>Any CP node selected

CPMERGE, *Lab*

Merges different couple sets with duplicate degrees of freedom into one couple set.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Degree of freedom label for coupled nodes (in the nodal coordinate system). Valid labels are: Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations) (in radians). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Fluid labels: PRES (pressure); VX, VY, or VZ (velocities). Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current). Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential); CURR (current). Diffusion label: CONC (concentration). The degree of freedom set is determined from all element types defined and the **DOF (p. 568)** command, if used.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Capacitor

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Curr Cntl CS

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Curr Cntl VS

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Diode

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Constant Amplitude

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Exponential

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Piecewise

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Pulse

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Curr Src>Sinusoidal

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Constant Amplitude

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Exponential

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Piecewise

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Pulse

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Indp Vltg Src>Sinusoidal

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Inductor

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mass Cond 2D

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mass Cond 3D

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Mutual Ind

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Resistor

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Strnd Coil

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Vltg Cntl CS

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Vltg Cntl VS

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Wire

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Electric>Zener Diode

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Linear Rotary

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Linear Trans

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Nonlin Rotary

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Nonlin Trans

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Damper>Slide Film

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Mass

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Linear Rotary

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Linear Trans

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Capacitor

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Constant Amplitude

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Exponential

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Piecewise

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Pulse

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Curr Src>Sinusoidal

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Constant Amplitude

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Exponential

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Piecewise

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Pulse

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Indp Vltg Src>Sinusoidal

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Inductor

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Resistor

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Piezoelectric>Wire

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

---

CPNGEN, *NSET*, *Lab*, *NODE1*, *NODE2*, *NINC*

Defines, modifies, or adds to a set of coupled degrees of freedom.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET

Set reference number (**CP (p. 421)**).

Lab

Degree of freedom label (**CP (p. 421)**).

NODE1, NODE2, NINC

Include in coupled set nodes *NODE1* to *NODE2* in steps of *NINC* (defaults to 1). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If -*NODE1*, delete range of nodes from set instead of including. A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Defines, modifies, or adds to a set of coupled degrees of freedom. May be used in combination with (or in place of) the **CP (p. 421)** command. Repeat **CPNGEN (p. 430)** command for additional nodes.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPSEL, *Type*, - - , - - , *VMIN*, *VMAX*, *VINC*

Selects coupled degree-of-freedom sets via predefined reference numbers.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set (default).

A

Select an additional set and add it to the current set.

U

Unselect a set from the current set.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

- -, - -

Unused fields.

The following fields are used only with *Type* = S, A, or U:

VMIN

Minimum value of coupled DOF reference number range.

VMAX

Maximum value of coupled DOF reference number range. *VMAX* defaults to *VMIN*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VINC

Value increment within the specified range. Defaults to 1.

Command Default

All coupled sets are selected.

Notes

The **CPSEL (p. 431)** command selects coupled degree-of-freedom sets (**CP (p. 421)**) via specified reference numbers. *VMIN*, *VMAX*, and *VINC* must be positive integer values.

For example, the following command selects a new set of coupled degree-of-freedom sets based on reference numbers 1 through 7:

```
CPSEL,S,,,1,7,1
```

Data are flagged as selected and unselected; no data are actually deleted from the database.

Use **CPLIST (p. 427)** to list coupled degree-of-freedom sets and their reference numbers. If a coupled degree-of-freedom set is selected but involves unselected nodes, that coupled degree-of-freedom set will not be listed by the **CPLIST (p. 427)** command, and the solver ignores it.

Internally coupled degrees of freedom are not affected by this command.

This command is also valid in POST1.

Menu Paths

This command cannot be accessed from a menu.

CPSGEN, *ITIME*, *INC*, *NSET1*, *NSET2*, *NINC*

Generates sets of coupled nodes from existing sets.

PREP7 (p. 22): Coupled DOF (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME*, *INC

Do this generation operation a total of *ITIME*s, incrementing all nodes in the existing sets by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

NSET1*, *NSET2*, *NINC

Generate sets from sets beginning with *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* is negative, *NSET2* and *NINC* are ignored and the last |*NSET1*| sets (in sequence from the maximum set number) are used as the sets to be repeated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates additional sets of coupled nodes (with the same labels) from existing sets. Node numbers between sets may be uniformly incremented.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Gen w/Same DOF

CQC,*SIGNIF*,*Label*,*--*,*ForceType*

Specifies the complete quadratic mode combination method.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single-point, multi-point, or DDAM response spectrum (*Spectype* = SPRS, MPRS, or DDAM on **SPOPT (p. 1837)**), the significance level of a mode is defined as the mode coefficient divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT (p. 1837)**,PSD.)

Label

Label identifying the combined mode solution output.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

--

Unused field.

ForceType

Label identifying the forces to be combined:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STATIC

Combine the modal static forces (default).

TOTAL

Combine the modal static plus inertial forces.

Notes

Damping is required for this mode combination method. The **CQC (p. 433)** command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine>CQC Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine>CQC Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine>CQC Method

Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine>CQC Method

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine>CQC Method

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine>CQC Method

---

*CREATE, *Fname*, *Ext*, - -

Opens (creates) a macro file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Do not use a directory path if file is to be read with the macro *Name* option of the ***USE (p. 2059)** command.

Ext

Filename extension (eight-character maximum).

Ext should not be used if file is to be read with the macro *Name* option of the ***USE (p. 2059)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- -

Unused field.

Notes

See the ***USE (p. 2059)** command for a discussion of macros. All commands following the ***CREATE (p. 434)** command, up to the ***END (p. 635)** command, are written to the specified file without being executed. An existing file of the same name, if any, will be overwritten. Parameter values are not substituted for parameter names in the commands when the commands are written to the file. Use ***CFWRITE (p. 318)** to create a file if this is desired. The resulting macro may be executed with a ***USE (p. 2059)** command (which also allows parameters to be passed into the macro) or a **/INPUT (p. 948)** command (which does not allow parameters to be passed in). Several macros may be stacked into a library file (***ULIB (p. 2051)**). You cannot use ***CREATE (p. 434)** within a DO loop.

This command is valid in any processor.

Menu Paths

Utility Menu>Macro>Create Macro

CS,*KCN*,*KCS*,*NORIG*,*NXAX*,*NXYPL*,*PAR1*,*PAR2*

Defines a local coordinate system by three node locations.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART

Cartesian

1 or CYLIN

Cylindrical (circular or elliptical)

2 or SPHE

Spherical (or spheroidal)

3 or TORO

Toroidal

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NORIG

Node defining the origin of this coordinate system. If *NORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NXAX

Node defining the positive x-axis orientation of this coordinate system.

NXYPL

Node defining the x-y plane (with *NORIG* and *NXAX*) in the first or second quadrant of this coordinate system.

PAR1

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local right-handed coordinate system by specifying three existing nodes: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system. See the **CLOCAL (p. 352)**, **CSKP (p. 438)**, **CSWPLA (p. 440)**, and **LOCAL (p. 1059)** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB (p. 1552)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>By 3 Nodes

---

CSCIR, *KCN*, *KTHET*, *KPHI*

Locates the singularity for non-Cartesian local coordinate systems.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Number of the local coordinate system in which singularity location is to be changed. Must be greater than 10.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KTHET

Theta singularity location for cylindrical, spherical, and toroidal systems:

0

Singularity at ±180°.

1

Singularity at 0° (360°).

KPHI

Phi singularity location for toroidal systems:

0

Singularity in phi direction at ±180°.

1

Singularity in phi direction at 0° (360°).

Command Default

Singularities at ±180°.

Notes

Continuous closed surfaces (circles, cylinders, spheres, etc.) have a singularity (discontinuity) at θ = ±180°. For local cylindrical, spherical, and toroidal coordinate systems, this singularity location may be changed to 0° (360°).

An additional, similar singularity occurs in the toroidal coordinate system at Φ = ±180° and can be moved with *KPHI*. Additional singularities occur in the spherical coordinate system at Φ = ±90°, but cannot be moved.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Move Singularity

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CSDELE, *KCN1*, *KCN2*, *KCINC*

Deletes local coordinate systems.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN1, KCN2, KCINC

Delete coordinate systems from *KCN1* (must be greater than 10) to *KCN2* (defaults to *KCN1*) in steps of *KCINC* (defaults to 1). If *KCN1* = ALL, *KCN2* and *KCINC* are ignored and all coordinate systems are deleted.

Notes

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node

Utility Menu>WorkPlane>Local Coordinate Systems>Delete Local CS

CSKP, *KCN*, *KCS*, *PORIG*, *PXAXS*, *PXYPL*, *PAR1*, *PAR2*

Defines a local coordinate system by three keypoint locations.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART

Cartesian

1 or CYLIN

Cylindrical (circular or elliptical)

2 or SPHE

Spherical (or spheroidal)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 or TORO

Toroidal

PORIG

Keypoint defining the origin of this coordinate system. If *PORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

PXAXS

Keypoint defining the positive x-axis orientation of this coordinate system.

PXYPL

Keypoint defining the x-y plane (with *PORIG* and *PXAXS*) in the first or second quadrant of this coordinate system.

PAR1

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local right-handed coordinate system by specifying three existing keypoints: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system. See the **CLOCAL (p. 352)**, **CS (p. 435)**, **CSWPLA (p. 440)**, and **LOCAL (p. 1059)** commands for alternate definitions. Local coordinate systems may be displayed with the **/PSYMB (p. 1552)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>By 3 Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CSLIST, *KCN1*, *KCN2*, *KCINC*

Lists coordinate systems.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN1*, *KCN2*, *KCINC

List coordinate systems from *KCN1* to *KCN2* (defaults to *KCN1*) in steps of *KCINC* (defaults to 1). If *KCN1* = ALL (default), *KCN2* and *KCINC* are ignored and all coordinate systems are listed.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Local Coord Sys

CSWPLA, *KCN*, *KCS*, *PAR1*, *PAR2*

Defines a local coordinate system at the origin of the working plane.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART

Cartesian

1 or CYLIN

Cylindrical (circular or elliptical)

2 or SPHE

Spherical (or spheroidal)

3 or TORO

Toroidal

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PAR1

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

PAR2

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines and activates a local right-handed coordinate system centered at the origin of the working plane. The coordinate system's local x-y plane (for a Cartesian system) or R-θ plane (for a cylindrical or spherical system) corresponds to the working plane. This local system becomes the active coordinate system. See the **CS (p. 435)**, **LOCAL (p. 1059)**, **CLOCAL (p. 352)**, and **CSKP (p. 438)** commands for alternate ways to define a local coordinate system. Local coordinate systems may be displayed with the **/PSYMB (p. 1552)** command.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>At WP Origin

---

CSYS, *KCN*

Activates a previously defined coordinate system.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Specifies the active coordinate system, as follows:

0 (default)

Cartesian

1

Cylindrical with global Cartesian Z as the axis of rotation

2

Spherical

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 or WP

Working Plane

5

Cylindrical with global Cartesian Y as the axis of rotation

6

Cylindrical with global Cartesian X as the axis of rotation

11 or greater

Any previously defined local coordinate system

Notes

The **CSYS (p. 441)** command activates a previously defined coordinate system for geometry input and generation. The **LOCAL (p. 1059)**, **CLOCAL (p. 352)**, **CS (p. 435)**, **CSKP (p. 438)**, and **CSWPLA (p. 440)** commands also activate coordinate systems as they are defined. To set the active element coordinate system attribute pointer, issue the **ESYS (p. 685)** command.

The active coordinate system for files created via the **CDWRITE (p. 293)** command is Cartesian (**CSYS (p. 441)**,0).

This command is valid in any processor.

CSYS (p. 441),4 (or **CSYS (p. 441)**,WP) activates working plane tracking, which updates the coordinate system to follow working plane changes. To deactivate working plane tracking, activate *any other* coordinate system (for example, **CSYS (p. 441)**,0 or **CSYS (p. 441)**,11).

CSYS (p. 441),5 is a cylindrical coordinate system with global Cartesian Y as the axis of rotation. The local x, y and z axes are radial, θ, and axial (respectively). The R-Theta plane is the global X-Z plane, as it is for an axisymmetric model. Thus, at $\theta = 0.0$, **CSYS (p. 441)**,5 has a specific orientation: the local x is in the global +X direction, local y is in the global -Z direction, and local z (the cylindrical axis) is in the global +Y direction.

CSYS (p. 441),6 is a cylindrical coordinate system with global Cartesian X as the axis of rotation. The local x, y and z axes are axial, radial, and θ (respectively). The R-Theta plane is the global Y-Z plane, as it is for an axisymmetric model. Thus, at $\theta = 0.0$, **CSYS (p. 441)**,6 has a specific orientation: the local x is in the global -Z direction, local y is in the global +Y direction, and local z (the cylindrical axis) is in the global +X direction.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node

Utility Menu>WorkPlane>Change Active CS to>Global Cartesian

Utility Menu>WorkPlane>Change Active CS to>Global Cylindrical

Utility Menu>WorkPlane>Change Active CS to>Global Spherical

Utility Menu>WorkPlane>Change Active CS to>Specified Coord Sys

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Utility Menu>WorkPlane>Change Active CS to>Working Plane

Utility Menu>WorkPlane>Offset WP to>Global Origin

/**CTYPE**, *KEY*, *DOTD*, *DOTS*, *DSHP*, *TLEN*

Specifies the type of contour display.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Type of display:

0

Standard contour display.

1

Isosurface display.

2

Particle gradient display.

3

Gradient triad display.

DOTD

Maximum dot density for particle gradient display (*KEY* = 2). Density is expressed as dots per screen width (defaults to 30).

DOTS

Dot size for particle gradient display (*KEY* = 2). Size is expressed as a fraction of the screen width (defaults to 0.0 (single dot width)).

DSHP

Spherical dot shape precision for particle gradient display (*KEY* = 2). (3D options are supported only on 3D devices):

0

Flat 2D circular dot.

1

Flat-sided 3D polyhedron.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

n

3D sphere with *n* (>1) polygon divisions per 90° of radius.

TLEN

Maximum length of triads for gradient triad display (*KEY* = 3). Value is expressed as a fraction of the screen width (defaults to 0.067).

Command Default

Standard contour display.

Notes

Use **/CTYPE (p. 443)**,STAT to display the current settings. Only the standard contour display (**/CTYPE (p. 443)**,0) and the isosurface contour display (**/CTYPE (p. 443)**,1) are supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Style

CURR2D

Calculates current flow in a 2D conductor.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

CURR2D (p. 444) invokes a macro which calculates the total current flowing in a conducting body for a 2D planar or axisymmetric magnetic field analysis. The currents may be applied source currents or induced currents (eddy currents). The elements of the conducting region must be selected before this command is issued. The total current calculated by the macro is stored in the parameter TCURR. Also, the total current and total current density are stored on a per-element basis in the element table (**ETABLE (p. 687)**) with the labels TCURR and JT, respectively. Use the **PLETAB (p. 1411)** and **PRETAB (p. 1490)** commands to plot and list the element table items.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Current

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CUTCONTROL, *Lab*, *VALUE*, *Option*

Controls time-step cutback during a nonlinear solution.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Specifies the criteria for causing a cutback. Valid labels are:

ALIMIT

Set the maximum edge-flux degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^{25}. This cutback trigger criterion is part of the physics limit set (p. 449).

CONCLIMIT

Set the maximum concentration degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^{25}. This cutback trigger criterion is part of the physics limit set (p. 449).

CRPLIMIT

Set values for calculating the maximum equivalent creep ratio allowed within a time step. If the calculated maximum creep ratio exceeds the defined creep-ratio limit, the program performs a cutback. For the first substep or the rebalance substeps after remeshing in nonlinear mesh adaptivity or rezoning, however, the user-defined maximum plastic-strain limit is ignored.

CUTBACKFACTOR

Key to set the cutback factor, a multiplier for reducing the time step interval to the fraction specified in *VALUE*, where 0 < *VALUE* < 1 (default is 0.5 if *VALUE* is not specified.) The option to change the cutback factor is valid only when automatic time stepping is on (**AUTOTS (p. 207)**,ON), which is the default.

DMGLIMIT

For damage models (such as generalized damage and anisotropic damage), the maximum allowable damage increment in a time step. If the calculated value exceeds *VALUE*, the program performs a cutback (bisection). If *VALUE* is unspecified, the program does not check the allowable damage increment.

DPPLMT

Set the maximum pore-pressure increment allowed within a time step. If the calculated maximum increment exceeds the specified limit, the program performs a cutback. This option has no default and is valid for coupled structural-pore-fluid-diffusion analysis only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DSPLIMIT

Set the maximum degree-of-freedom increment allowed within a time step. Considering all degrees of freedom at every node, if the absolute value of the maximum incremental degree-of-freedom solution within in a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^7.This cutback trigger criterion is the sole member of the DSPLIMIT set (p. 449).

EMFLIMIT

Set the maximum electromotive force degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^{25}. This cutback trigger criterion is part of the physics limit set (p. 449).

MAGLIMIT

Set the maximum scalar magnetic potential degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^{25}. This cutback trigger criterion is part of the physics limit set (p. 449).

MDMG

For regularized microplane damage models, the maximum allowable microplane homogenized damage increment in a time step. If the calculated value exceeds *VALUE*, the program performs a cutback (bisection). If *VALUE* is unspecified, the program does not check the allowable microplane homogenized damage increment.

PLSLIMIT

Maximum equivalent plastic strain allowed within a time-step (substep). If the calculated value exceeds *VALUE*, the program performs a cutback (bisection). Default: *VALUE* = 0.15 (15 percent)

If **CUTCONTROL (p. 445)** with *Lab* = PLSLIMIT is not issued, the minimum time step specified is reached, and the maximum plastic limit calculated from the solution exceeds 15 percent, the program generates a warning and continues the Newton iterations.

If **CUTCONTROL (p. 445)** with *Lab* = PLSLIMIT is issued, the minimum time step specified is reached, and the maximum plastic limit calculated exceeds the specified limit, the program generates an error and stops the Newton iterations. For the first substep or the rebalance substeps after remeshing in nonlinear mesh adaptivity or rezoning, however, the user-defined maximum plastic-strain limit is ignored.

PRESLIMIT

Set the maximum pressure degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^{25}. This cutback trigger criterion is part of the physics limit set (p. 449).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ROTLIMIT

Set the maximum rotation degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^7. This cutback trigger criterion is part of the physics limit set (p. 449).

TEMPLIMIT

Set the maximum temperature degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^{25}. This cutback trigger criterion is part of the physics limit set (p. 449) and is also valid for coupled structural-pore-fluid-diffusion-thermal analysis.

ULIMIT

Set the maximum translation degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^7. This cutback trigger criterion is part of the physics limit set (p. 449).

VOLTLIMIT

Set the maximum volt degree-of-freedom increment allowed within a time step. If the absolute value of the calculated increment within a time step exceeds *VALUE*, the program performs a cutback. If not specified, default *VALUE* = 1.0 x 10^{25}. This cutback trigger criterion is part of the physics limit set (p. 449).

VSLIMIT

For viscoelastic materials, the maximum equivalent viscous strain increment allowed within a time step. If the calculated value exceeds *VALUE* (default = 0.01), the program performs a cutback (bisection). If *VALUE* = 0, the program does not check the equivalent viscous strain increment.

Other Valid Labels:

NOITERPREDICT

If *VALUE* = 0 (default), the program predicts the number of iterations for nonlinear convergence and performs a cutback earlier than the number of iterations specified (**NEQIT (p. 1214)**). This is the recommended option.

If *VALUE* = 1, the solution iterates (if nonconvergent) to **NEQIT (p. 1214)** number of iterations before a cutback is invoked. It is sometimes useful for poorly-convergent problems, but rarely needed in general.

Bisection is also controlled by contact status change, plasticity or creep strain limit, and other factors. If any of these factors occur, bisection still occurs, regardless of the NOITERPREDICT setting.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NPOINT

Number of points in a cycle for a second order dynamic equation, used to control automatic time-stepping. If the number of solution points per cycle is less than *VALUE*, the program performs a cutback in time step size. Default: *VALUE* = 13 (linear analysis) or 5 (nonlinear analysis). A larger number of points yields a more accurate solution but also increases the solution run time.

This option works well for linear problems. For nonlinear analyses, other factors such as contact status changes and solution convergence rate can overwrite NPOINT. (See Automatic Time-Stepping in the *Theory Reference* for more information about automatic time-stepping.)

VALUE

Numeric value for the specified cutback criterion. For *Lab* = CRPLIMIT only, *VALUE* is the creep criteria for the creep ratio limit.

Option

Type of creep analysis. Valid for *Lab* = CRPLIMIT only.

IMPRATIO

Set the maximum creep ratio value for implicit creep. Default = 0.0 (no creep limit control). Any positive value is valid.

STSLIMIT

Stress threshold for calculating the creep ratio. For integration points with effective stress below this threshold, the creep ratio does not cause cutback. Default = 0.0. Any positive value is valid.

STNLIMIT

Elastic strain threshold for calculating the creep ratio. For integration points with effective elastic strain below this threshold, the creep ratio does not cause cutback. Default = 0.0. Any positive value is valid.

Notes

A cutback is a method for automatically reducing the step size when either the solution error is too large or the solution encounters convergence difficulties during a nonlinear analysis.

If a convergence failure occurs, the program reduces the time step interval to a fraction of its previous size and automatically continues the solution from the last successfully converged time step. If the reduced time step again fails to converge, the program again reduces the time step size and proceeds with the solution. This process continues until convergence is achieved or the minimum specified time step value is reached.

A cutback occurs when a trigger criteria is encountered. The magnitude of the time-step reduction is determined by the cutback factor. Both the trigger and the cutback factor can be specified via **CUTCONTROL (p. 445)**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The trigger criteria can be specified for individual physics as detailed in *Lab* (p. 445) descriptions above.
- The cutback factor is 0.5 by default, and can be specified via the CUTBACKFACTOR (p. 445) label described above.

CUTCONTROL (p. 445) can be issued multiple times.

DSPLIMIT Set

DSPLIMIT is the sole label in the DSPLIMIT Set. It can be used to specify the cutback trigger criterion as a limit on the maximum degree-of-freedom increment within a time step, considering all degrees of freedom in an analysis (see DSPLIMIT label description (p. 446) above).

Physics Limit Set

The physics limit set comprises the following labels: ULIMIT, ROTLIMIT, TEMPLIMIT, PRESLIMIT, VOLTLIMIT, EMFLIMIT, MAGLIMIT, ALIMIT, CONCLIMIT. They can be used to specify distinct degree-of-freedom limits as cutback triggers for individual physics in a multiphysics analysis. Whenever one of these distinct limits is encountered, a cutback is triggered.

Degree-of-freedom limits as cutback trigger criteria are specified via **CUTCONTROL (p. 445)** with labels from either the DSPLIMIT set or the physics limit set. If multiple **CUTCONTROL (p. 445)** commands include labels from both of these sets, the last command issued determines whether the cutback trigger criteria is set by the DSPLIMIT or the physics limit set (see example command listing snippets (p. 450) below).

Command Default: If no **CUTCONTROL (p. 445)** commands are issued with labels from the DSPLIMIT set or the physics limit set, the default cutback trigger criterion is DSPLIMIT with *VALUE* = 1.0×10^{7}.

Creep and Viscoelastic Analyses

For creep and viscoelastic analyses, the cutback procedure is similar; the process continues until the minimum specified time step size is reached. However, if the criterion is exceeded, the program issues a warning but continues the substep until the analysis is complete. In this case, convergence is achieved but the criterion is not satisfied.

CUTCONTROL (p. 445) enables a step size reduction for analyses experiencing convergence difficulties; **OPNCONTROL (p. 1326)** is an analogous but opposite command that increases the step size to speed up converging analyses.

Example Usage

Friction Stir Welding Simulation (specify PLSLIMIT on CUTCONTROL)

Wire Crimping Simulation (specify PLSLIMIT on CUTCONTROL)

Control the maximum incremental pore pressure allowed in a time step via DPPLIMIT on CUTCONTROL

Regularized Microplane Damage Models (specify MDMG on CUTCONTROL)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multiple CUTCONTROL (p. 445) Commands with labels from both the DSPLIMIT and the Physics Limit Sets

When multiple **CUTCONTROL (p. 445)** commands include labels from both the DSPLIMIT and the physics limit sets, DSPLIMIT is ignored unless *Lab* = DSPLIMIT on the last **CUTCONTROL (p. 445)** command issued as illustrated in the examples below.

- **Label on the last CUTCONTROL (p. 445) command issued is from the physics limit set**: The cutback trigger criteria are those specified by the physics limit set, and the earlier **CUTCONTROL (p. 445)**,DSPLIMIT command is ignored.

```
CUTCONTROL,TEMPLIMIT,10
CUTCONTROL,DSPLIMIT,0.7
CUTCONTROL,ULIMIT,0.2   !trigger criteria is specified by physics limit set (and DSPLIMIT is ignored)
!both ULIMIT and TEMPLIMIT contribute to the physics limit set of cutback trigger criteria
```

- ***Lab* = DSPLIMIT on the last CUTCONTROL (p. 445) command issued**: DSPLIMIT sets the cutback trigger criterion and earlier **CUTCONTROL (p. 445)** command(s) issued with *Lab* = labels from the physics limit set are ignored.

```
CUTCONTROL,TEMPLIMIT,10
CUTCONTROL,ULIMIT,0.2
CUTCONTROL,DSPLIMIT,0.7 !trigger criteria is specified by DSPLIMIT (and physics limit set labels are ignored)
```

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Cutback Control

Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Solution>Load Step Opts>Nonlinear>Cutback Control

/CVAL, *WN*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*, *V7*, *V8*

Specifies nonuniform contour values on stress displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

V1, V2, V3, . . . , V8

Up to 8 contour values may be specified (in ascending order). The 0.0 value (if any) must not be the last value specified. If no values are specified, all contour specifications are erased and contours are automatically calculated.

Command Default

Nine contour values uniformly spaced between the extreme values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is similar to the **/CONTOUR (p. 415)** command. With **/CVAL (p. 450)**, however, you define the upper level of each contour band instead of having the contours uniformly distributed over the range. The minimum value (including a zero value for the first band) for a contour band cannot be specified. If you use both **/CONTOUR (p. 415)** and **/CVAL (p. 450)**, the last command issued takes precedence.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Non-uniform Contours

CVAR, *IR*, *IA*, *IB*, *ITYPE*, *DATUM*, *Name*

Computes covariance between two quantities.

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previous variable, the previous variable will be overwritten with this result.

IA*, *IB

Reference numbers of the two variables to be operated on. If only one, leave *IB* blank.

ITYPE

Defines the type of response PSD to be calculated:

0,1

Displacement (default).

2

Velocity.

3

Acceleration.

DATUM

Defines the reference with respect to which covariance is to be calculated:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Absolute value.

2

Relative to base (default).

Name

Thirty-two character name for identifying the variable on listings and displays. Embedded blanks are compressed upon output.

Notes

This command computes the covariance value for the variables referenced by the reference numbers *IA* and *IB*. If *DATUM* = 2, the variable referenced by *IR* will contain the individual modal contributions (that is, the dynamic or relative values). If *DATUM* = 1, the variable referenced by *IR* will contain the modal contributions followed by the contributions of pseudo-static and covariance between dynamic and pseudo-static responses. `File.psd` must be available for the calculations to occur.

Menu Paths

Main Menu>TimeHist Postpro>Calc Covariance

/CWD, *DIRPATH*

Changes the current working directory.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DIRPATH

The full path name of the new working directory.

Notes

After issuing the /CWD command, all new files opened with no default directory specified (via the **FILE (p. 752)**, **/COPY (p. 416)**, or **RESUME (p. 1601)** commands, for example) default to the new *DIRPATH* directory.

Menu Paths

Utility Menu>File>Change Directory

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CYCCALC,*FilePrefix*,*FileFormat*,*Separator*

Calculates results from a cyclic harmonic mode-superposition analysis using the specifications defined by CYCSPEC (p. 475).

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FilePrefix

Each result table (corresponding to each **CYCSPEC (p. 475)** specification) is written to a file beginning with *FilePrefix*. If blank (default), the result tables are written to the output file.

FileFormat

If *FilePrefix* is specified, then use *FileFormat* to specify the format of the file to be written:

FORM

Formatted file (default)

CSV

Comma-separated value file

Separator

If *FileFormat* is CSV, use *Separator* to specify the field separator:

COMMA

Use a comma (,) as the field separator (default)

COLON

Use a colon (:) as the field separator

DOT

Use a period (.) as the field separator

Command Default

Write the result tables to the output file.

Notes

CYCCALC (p. 453) loops through the specification given by **CYCSPEC (p. 475)** and computes the requested outputs. The outputs are given in a table format, with the rows corresponding to each frequency solution from the harmonic analysis, and the columns corresponding to each sector. The table entries are the maximum value of the specified quantity at the specified location in the sector. In addition, columns containing the maximum value at the frequency, the sector in which it occurs, and the node in the sector at which it occurs are output.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *FilePrefix* is specified, a file is created for each output table with the name *FilePrefix_node_type.ext*, where *node* is the node number or component name, *type* is the item/component requested, and the file extension *.ext* is either .txt or .csv, depending on *FileFormat*.

A **SET (p. 1724)** command must precede the **CYCCALC (p. 453)** command.

The **CYCCALC (p. 453)** results are based on the currently active **RSYS (p. 1639)**, **SHELL (p. 1784)**, **LAYER (p. 1015)**, and **AVPRIN (p. 210)** settings.

The **CYCCALC (p. 453)** command only supports matched nodes. For more details on matching cyclic edge node pairs see Edge Component Pairs in the *Cyclic Symmetry Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

/CYCEXPAND, *WN*, *OPTION*, *Value1*, *Value2*

Graphically expands displacements, stresses and strains of a cyclically symmetric model.

PREP7 (p. 22): Special Purpose (p. 37)
POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

The window number to which the expansion applies. Valid values are 1 through 5. The default value is 1. The window number applies only to the AMOUNT argument.

OPTION

One of the following options:

ON

Activates cyclic expansion using the previous settings (if any). If no previous settings exist, this option activates the default settings. This option is default.

DEFAULT

Resets cyclic expansion to the default settings.

OFF

Deactivates cyclic expansion.

STATUS

Lists the current cyclic expansion settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AMOUNT

The number of repetitions *or* the total angle.

Value1

NREPEAT

Value2

The number of repetitions. The default is the total number of sectors in 360 degrees.

or

Value1

ANGLE

Value2

The total angle in degrees. The default is 360.

WHAT

A specified portion or subset of the model to expand:

Value1

The component name of the elements to expand. The default is all selected components.

EDGE

Sector edge display key. Possible *Value1* settings are:

-1

Suppresses display of edges between sectors even if the cyclic count varies between active windows. This setting is not valid for cyclic mode-superposition (MSUP) harmonic analyses.

Caution:

Plots with fewer than the maximum number of repetitions may have missing element faces at the sector boundaries.

0 or OFF

Averages stresses or strains across sector boundaries. This value is the default (although the default reverts to 1 or ON if the cyclic count varies between active windows).

1 or ON

No averaging of stresses or strains occurs, and sector boundaries are shown on the plot. This setting is not valid for cyclic MSUP harmonic analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PHASEANG

Possible *Value1* settings are:

n

The phase angle shift in degrees. The valid range for *n* is 0 through 360. The default is 0. For a modal solution, this value is typically the phase angle obtained via the **CYCPHASE (p. 472)** command. The expanded modal results are printed or displayed for the specified phase angle shift.

AMPLITUDE (or *n* $\geq$ 360)

The amplitude is reported, except for the following circumstances where the amplitude solution is not valid:

- non-component results (such as equivalent stress)
- modal analyses (no amplitude is calculated, and the expanded modal results are printed or displayed at a phase angle of 360°).

SWEEP

For a mode-superposition harmonic solution, the maximum values across a phase angle sweep are reported.

Notes

In preprocessing, the **/CYCEXPAND (p. 454)** command verifies a cyclically symmetric model by graphically expanding it partially or through the full 360 degrees.

For the postprocessing plot nodal solution (**PLNSOL (p. 1425)**) operation, the command graphically expands displacements, stresses and strains of a cyclically symmetric model partially or though the full 360 degrees by combining the *real* (original nodes and elements) and *imaginary* (duplicate nodes and elements) parts of the solution.

For the print nodal solution (**PRNSOL (p. 1505)**) operation, the command expands the printed output of displacements or stresses on a sector-by-sector basis. To learn more about specific **PRNSOL (p. 1505)** behaviors in cyclic analyses, see Using the /CYCEXPAND Command in the *Cyclic Symmetry Analysis Guide*.

Use of the **/CYCEXPAND (p. 454)** command does *not* change the database. The command does not modify the geometry, nodal displacements or element stresses.

The command affects element and result plots only. It has no effect on operations other than plot element solution (**PLESOL (p. 1401)**), plot nodal solution (**PLNSOL (p. 1425)**), print nodal solution (**PRNSOL (p. 1505)**), and calculate harmonic solution (**CYCCALC (p. 453)**). Operations other than **PLESOL (p. 1401)**, **PLNSOL (p. 1425)**, **PRNSOL (p. 1505)**, or **CYCCALC (p. 453)** work on the *unprocessed* real and imaginary parts of a cyclic symmetry solution

If you issue a **/CYCEXPAND (p. 454)**,,0FF command, you cannot then expand the model by simply issuing another **/CYCEXPAND (p. 454)** command (for example, to specify an NREPEAT value for the number of repetitions). In such a case, you must specify **/CYCEXPAND (p. 454)**,,0N, which activates expansion using the previous settings (if any) or the default settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The command requires PowerGraphics and will turn PowerGraphics on (**/GRAPHICS (p. 866)**,POWER) if not already active. Any setting which bypasses PowerGraphics (for example, **/PBF (p. 1360)**) also bypasses cyclic expansion; in such cases, the **/CYCEXPAND (p. 454)** command displays unprocessed real and imaginary results.

The **CYCPHASE (p. 472)** command uses full model graphics (**/GRAPHICS (p. 866)**,FULL) to compute peak values. Because of this, there may be slight differences between max/min values obtained with **CYCPHASE (p. 472)**, and those obtained via **/CYCEXPAND (p. 454)**, which uses power graphics (**/GRAPHICS (p. 866)**,POWER).

For PHASEANG = AMPLITUDE (or 360) with a cyclic full harmonic solution, the only appropriate coordinate system is the solution coordinate system (**RSYS (p. 1639)**,SOLU).

Load case operations (**LCOPER (p. 1026)**) are not valid during a cyclic expansion using **/CYCEXPAND (p. 454)**.

To learn more about analyzing a cyclically symmetric structure, see the *Cyclic Symmetry Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Cyclic Analysis>Cyc Expansion

Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyc Expansion

CYCFILES,*FnameRst*,*ExtRst*,*FnameRfrq*,*ExtRfrq*

Specifies the data files where results are to be found for a cyclic symmetry mode-superposition harmonic analysis.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FnameRst

The file name and directory path of the results file from the cyclic modal solution. Defaults to Jobname.

ExtRst

File name extension for *FnameRst*. Defaults to rst.

FnameRfrq

The file name and directory path of the results file from the cyclic mode-superposition harmonic solution. Defaults to the value of the *FnameRst* argument.

ExtRfrq

File name extension for *FnameRfrq*. Defaults to rfrq.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

No defaults are available for the **CYCFILES (p. 457)** command. You must issue this command to properly postprocess the results of a cyclic symmetry mode-superposition harmonic analysis. If issued with no arguments, the postprocessing will be done using `Jobname.rst` and `Jobname.rfrq` from the current working directory.

Menu Paths

This command cannot be accessed from a menu.

CYCFREQ, *Option*, *Value1*, *Value2*, *Value3*, *Value4*, *Value5*

Specifies solution options for a cyclic symmetry mode-superposition harmonic analysis.

PREP7 (p. 22): Special Purpose (p. 37)
SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

One of the following options:

AERO

Specify the array containing the aerodynamic damping coefficients.

Value1

The name of the array containing the aerodynamic stiffness damping coefficients.

BLADE

Blade information required for a mistuning or aerodamping analysis.

Value1

The name of the nodal component containing the blade boundary nodes at the blade-to-disk interface. Also include boundary nodes at any shroud interfaces.

Value2

The name of the element component containing the blade elements.

Value3

The number of blade modes to include in the CMS reduction.

Value4

The lower bound of the frequency range of interest. This value is optional.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value5

The upper bound of the frequency range of interest. This value is optional.

DEFAULT

Set the default cyclic harmonic solution settings.

EO

Excitation engine order.

Value1

An integer value indicating the the excitation order. The loadings on the other sectors will be related to the loading on the base sector based on the engine order phase shift.

Value2

The name of the Mechanical APDL array containing the modal forces corresponding to the modes kept in the mode-superpostion analysis.

MIST

Mistuning parameters.

Value1

The type of mistuning:

K

Stiffness (frequency) mistuning

Value2

The name of the array containing the stiffness mistuning parameters.

MODAL

Specifies if a damped modal analysis should be performed on the reduced system.

Value1

On/Off key.

0 (OFF or NO)

No modal solution. Perform the harmonic solution.

1 (ON or YES)

Perform a damped modal analysis of the reduced system in order to obtain the complex frequencies. The harmonic solution is not performed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value2

Number of modes for the damped modal analysis.

Value3

The beginning, or lower end, of the frequency range of interest (in Hz).

Value4

The ending, or upper end, of the frequency range of interest (in Hz).

RESTART

Defines the point at which to restart the harmonic analysis.

Value1

The restart point:

OFF

No restart (default)

SWEEP

Restart for a new frequency sweep range (**HARFRQ (p. 885)**)

MIST

Restart for new mistuning parameters (new mistuning arrays)

USER

Causes the program to call for a user-defined solution.

Value1-5

Values passed down to the user-defined solution.

STATUS

List the harmonic solution option settings active for the cyclic model.

Command Default

No defaults are available for the **CYCFREQ (p. 458)** command. You must specify an *Option* label when issuing this command. Other values which may be necessary depend upon which *Option* label you specify.

Notes

The program solves a cyclically symmetric model (set up via the **CYCLIC (p. 462)** command during pre-processing) at the harmonic indices specified via the **CYCOPT (p. 466)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When *Option* = AERO, the aerodynamic coefficients are specified in a 5×(N×r) array (***DIM (p. 530)**), where N is the number of blades and r can be any positive integer. Each column has the structure:

$$i,m,n,V_{\text{real}},V_{\text{imag}}$$

where:

i = the i[th] interblade phase angle (IBPA)

m = the m[th] vibrating blade mode

n = the n[th] blade mode generating the pressure oscillations

V_{real} and V_{imag} = the real and imaginary coefficients.

One aerodynamic damping coefficient must be specified for each IBPA (equal to the number of blades) while keeping m and n constant.

The following table shows how the IBPA index (i) relates to other quantities for a system with 22 blades:

IBPA Index (i)	Nodal Diameter	IBPA (degrees)
0	0	0
1	1	16.36
⋮	⋮	⋮
20	-2	-32.73
21	-1	-16.36

The **CYCFREQ (p. 458)**,AERO command is only valid if **CYCFREQ (p. 458)**,BLADE is also specified. The blade mode numbers, m and n, are relative to the values kept in the **CYCFREQ (p. 458)**,BLADE command.

For constant (frequency-independent) mistuning, the stiffness parameters are specified in an N×1 array (***DIM (p. 530)**) where N is the number of blades.

For stiffness mistuning, each row entry represents the deviation of Young's modulus from nominal, $E_n=\left(1+\delta_n\right)E_0$ (or equivalently, the ratio of the frequency deviation squared). Each frequency can also be independently mistuned, in which case the array is N×M, where M is the number of blade frequencies (*Value3* of **CYCFREQ (p. 458)**,BLADE). The entries in each row therefore correspond to the ratio of the mistuned frequency to the tuned frequency squared minus one:

$$\delta f_i^n=\left(\frac{\bar{\omega}_i^n}{\omega_i^n}\right)^2-1$$

The USER option activates the solution macro CYCMSUPUSERSOLVE.MAC. The normal solution is skipped. You may implement your own mistuning solution using APDL and APDL Math operations, or call your own program for the solution.

The **CYCFREQ (p. 458)** command is valid in the preprocessing and solution stages of an analysis.

The **CYCFREQ (p. 458)**,MODAL,ON command writes modal frequencies to the output file. No other postprocessing is available for this modal solve.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using **CYCFREQ (p. 458)**,RESTART, only mistuning parameters or frequency range may be changed. All other changes in parameters are ignored. This type of restart can only be performed by exiting the current mistuning solution using **FINISH (p. 758)** and re-entering the solution phase using **/SOLU (p. 1821)** and then calling the desired **CYCFREQ (p. 458)**,RESTART command.

To learn more about analyzing a cyclically symmetric structure, see the *Cyclic Symmetry Analysis Guide*.

Example Usage

Analysis and Solution Controls section of the Technology Showcase Example Problem: Forced-response analysis of a mistuned bladed disk with aerodamping.

For an example demonstrating how to apply modal loads directly in the cyclic harmonic analysis step of a mode-superposition harmonic cyclic symmetry analysis, see Apply Modal Loads Directly Using an Ordered Array in the *Cyclic Symmetry Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

---

*CYCLE

Bypasses commands within a do-loop.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Bypasses all commands between this command and the ***ENDDO (p. 636)** command within a do-loop. The next loop (if applicable) is initiated. The cycle option may also be conditionally executed [Use the ***IF (p. 927)**]. The ***CYCLE (p. 462)** command must appear on the same file as the ***DO (p. 567)** command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

CYCLIC, *NSECTOR,ANGLE,KCN ,Name,USRCOMP, UsrNMap*

Specifies a cyclic symmetry analysis.

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSECTOR

The number of sectors in the full 360 degrees, *or* one of the following options:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STATUS

Indicates the current cyclic status.

OFF

Resets model to normal (non-cyclic) status and removes the duplicate sector if it exists. This option also deletes automatically detected edge components (generated when *USRCOMP* = 0).

UNDOUBLE

Removes the duplicate sector if it exists. The duplicate sector is created during the solution (**SOLVE (p. 1822)**) stage of a modal cyclic symmetry analysis.

Note:

The duplicate sector is necessary for displaying cyclic symmetry analysis results during postprocessing (**/POST1 (p. 1461)**).

If you specify a value of STATUS, OFF or UNDOUBLE, the command ignores all remaining arguments.

ANGLE

The sector angle in degrees.

KCN

An arbitrary reference number assigned to the cyclic coordinate system. The default value of 0 specifies automatic detection.

Name

The root name of sector low- and high-edge components (line, area, or node components). The default root name (when *USRCOMP* = 0) is "CYCLIC". A root name that you specify can contain up to 11 characters.

The naming convention for each low- and high-edge component pair is either of the following:

*Name*_m*xx*l, *Name*_m*xx*h (potentially matched node patterns)

*Name*_u*xx*l, *Name*_u*xx*h (potentially unmatched node patterns)

The *Name* value is the default ("CYCLIC ") or specified root name and *xx* is the component pair ID number (sequential, starting at 01).

USRCOMP

The number of pairs of user-defined low- and high-edge components on the cyclic sector (if any). The default value of 0 specifies automatic detection of sector edges; however, the automatic setting is not valid in all cases. (For more information, see the Notes section below.) If the value is greater than 0, no verification of user-defined components occurs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UsrNMap

The name of a user-defined array specifying the matching node pairs between the sector low and high edges. Valid only when *USRCOMP* = 0. Skips the automatic detection of sector edges. Node pairs may be input in any order, but the low edge node must be the first entry in each pair.

```
*DIM,MYMAP,ARRAY,2,14 ! specifying 14 low-high edge node pairs
*set,mymap(1, 1), 107, 108 ! low node 107 <> high node 108
*set,mymap(1, 2), 147, 211 ! low node 147 <> high node 211
*set,mymap(1, 3), 110, 109 ! low node 110 <> high node 109
        ! etc for node pairs 4 through 14
cyclic,12,,1,,,MYMAP  ! use array MYMAP to generate cyclic CEs
```

Command Default

The default **CYCLIC (p. 462)** command (issuing the command with no arguments) detects the number of sectors (*NSECTOR*), the sector angle (*ANGLE*), and the coordinate system (*KCN*) based upon the existing solid or finite-element model. The command also detects sector low- and high-edge components in most cases and assigns the default root name " CYCLIC" to the components.

Notes

You can input your own value for *NSECTOR*, *ANGLE* or *KCN*; if you do so, the command verifies argument values before executing.

When *USRCOMP* = 0 and *UsrNMap* = blank (default), the **CYCLIC (p. 462)** command automatically detects low- and high-edge components for models that consist of any combination of line, area, or volume elements. If a solid model exists, however, the command uses only the lines, areas, and/or volumes to determine the low- and high-edge components; the elements, if any, are ignored.

Nodes will be automatically rotated unless **CYCOPT (p. 466)**,USRROT,YES has been specified.

If you issue a **CYCOPT (p. 466)**,TOLER command to set a tolerance for edge-component pairing *before* issuing the **CYCLIC (p. 462)** command, the **CYCLIC (p. 462)** command uses the specified tolerance when performing automatic edge-component detection.

For 2D models, autodetection does not consider the **CSYS (p. 441)**,5 or **CSYS (p. 441)**,6 coordinate system specification. Autodetection for 180 degree (two-sector) models is not possible unless a central hole exists.

The **CYCLIC (p. 462)** command sets values and keys so that, if possible, the area-mesh (**AMESH (p. 121)**) or volume-mesh (**VMESH (p. 2140)**) command meshes the sector with matching node and element face patterns on the low and high edges. (The command has no effect on any other element-creation command.)

Issue the **CYCLIC (p. 462)** command *prior* to the meshing command to, if possible, produce a mesh with identical node and element patterns on the low and high sector edges. Only the **AMESH (p. 121)** or **VMESH (p. 2140)** commands can perform automated matching. (Other meshing operation commands such as **VSWEEP (p. 2168)** cannot.) If you employ a meshing operation other than **AMESH (p. 121)** or **VMESH (p. 2140)**, you should ensure that node and element face patterns match, if desired. The **CYCLIC (p. 462)** command output indicates whether each edge-component pair has or can produce a matching node pair.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A cyclic solution (via the **SOLVE (p. 1822)** command) allows dissimilar mesh patterns on the extreme boundaries of a cyclically symmetric model. The allowance for dissimilar patterns is useful when you have only finite-element meshes for your model but not the geometry data necessary to remesh it to obtain identical node patterns. In such cases, it is possible to obtain solution results, although perhaps at the expense of accuracy. A warning message appears because results may be degraded near the sector edges.

The constraint equations (CEs) that tie together the low and high edges of your model are generated at the solution stage of the analysis from the low- and high-edge components (and nowhere else). You should verify that automatically detected components are in the correct locations and that you can account for all components; to do so, you can list (**CMLIST (p. 366)**) or plot (**CMPLOT (p. 371)**) the components.

If you issue the **CYCLIC (p. 462)** command after meshing and have defined element types with rotational degrees of freedom (DOFs), Mechanical APDL generates cyclic CEs for rotational DOFs that may not exist on the sector boundaries. Issue **CYCOPT (p. 466)**,DOF to prevent unused rotational terms from being generated.

Modal cyclic symmetry analysis is supported by the following eigensolvers:

- Block Lanczos (**MODOPT (p. 1140)**,LANB)
- PCG Lanczos (**MODOPT (p. 1140)**,LANPCG)
- Super Node (**MODOPT (p. 1140)**,SNODE)
- Subspace (**MODOPT (p. 1140)**,SUBSP)

To learn more about analyzing a cyclically symmetric structure, see the *Cyclic Symmetry Analysis Guide*.

When using the **CYCLIC (p. 462)** command to automatically detect the sector, if an area is defined with the **AL (p. 112)** command, the lines need to be oriented to form the closed curve.

Menu Paths

Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>Auto Defined

Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>Status

Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Model>User Defined

Main Menu>Preprocessor>Modeling>Cyclic Sector>Del Dupl Sector

Main Menu>Preprocessor>Modeling>Cyclic Sector>Reset (OFF)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CYCOPT, *OPTION*, *Value1*, *Value2*, *Value3*, *Value4*, *Value5*, *Value6*, *Value7*

Specifies solution options for a cyclic symmetry analysis.

PREP7 (p. 22): Special Purpose (p. 37)
SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

OPTION

Cyclic symmetry analysis option. There is no default. You must choose one of the following options:

BCMULT

Controls whether cyclic sector array parameter names are reused or created new for multiple entities.

Value1

The flag value.

0 (OFF or NO)

Create new array parameter names (default)

1(ON or YES)

Reuse array parameter names

COMBINE

For linear static cyclic symmetry analysis with non-cyclically symmetric loading only, expands and combines all harmonic index solutions and writes them to the results file during the solution phase of the analysis.

Value1

The flag value.

0 (OFF or NO)

Disable combining of harmonic index solutions (default)

1 (ON or YES)

Enable combining of harmonic index solutions

DEFAULT

Set the default cyclic solution settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DOF

The degrees of freedom to couple from the nodes on the low sector boundary to nodes on the high boundary:

Value1

The component pair ID number.

Value2, Value3, Value4, ..., Value7

The constraint-equation/-coupling degree of freedom (DOF) for this pair. Repeat the command to add other DOFs. The default is constraint-equation/-coupling *all* applicable DOFs.

FACETOL

Tolerance for inclusion of surface nodes into your base sector. Autodetect defaults to 15°, accommodating most sections. Specify a new *Value1* only when extreme cut angles or complex model geometry cause surface nodes to be excluded. See Notes (below) for more information.

Ansys, Inc. recommends that successful auto-detection depends more on the value of ANGTOL than the value of FACETOL. Please refer to **CYCOPT** Auto Detection Tolerance Adjustments for Difficult Cases for more information about auto-detection and the **CYCOPT (p. 466)** command.

Value1

The face tolerance applies only to auto detection from node/element models (already meshed and no solid model), and it defaults to 15°.

HINDEX

The harmonic index ranges to be solved. Applies to static and harmonic analyses only if *Value5* = STATIC (p. 468).

By default, the **SOLVE (p. 1822)** command loops through *all* available harmonic indices. Static and harmonic analyses only solve harmonic indices required for the applied loads. All other analyses solve them all.

EVEN / ODD

For low-frequency electromagnetic analysis only, EVEN specifies a symmetric solution and ODD specifies an antisymmetric solution.

The value you specify is based on the harmonic index: EVEN (default) indicates harmonic index = 0, and ODD indicates harmonic index = *N* / 2 (where *N* is an integer representing the number of sectors in 360°). A value of ODD applies only when *N* is an even number.

The **CYCOPT (p. 466)** command with this HINDEX option is cumulative. To remove an option (for example, EVEN), issue this command: **CYCOPT (p. 466)**,HINDEX,EVEN,,,-1

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Solve all applicable harmonic indices.

Note:

Value2 must be blank.

Value1*, *Value2*, *Value3

Solve harmonic indices in range *Value1* through *Value2* in steps of *Value3*. Repeat the command to add other ranges. The default solves *all* applicable harmonic indices.

Value4

The only valid value is -1. If specified, it removes *Value1* through *Value2* in steps of *Value3* from the set to solve. By default, if *Value4* = -1 then *Value1* = 0, *Value2* = 0, and *Value3* = 1.

Value5

For static and harmonic analyses, enter either a number or STATIC:

If *Value5* = <*number*>

the number entered sets the tolerance for determining if the Fourier contribution of a load is significant (default = 1.0E-5). In this case, the harmonic index selection is disabled.

If *Value5* = STATIC

the harmonic index selection is enabled, and no verification is done on the significance of unselected harmonic indicies.

Note:

Improper selection of the harmonic index range may produce incomplete (incorrect) results.

LDSECT

Restricts subsequently defined force loads and surface loads to a specified sector. The restriction remains in effect until you change or reset it. This option is not available for harmonic analyses based on mode-superposition (**CYCOPT (p. 466)**,MSUP,1)

Value1

The sector number. A value other than 0 (default) is valid for a cyclic symmetry analysis with non-cyclically symmetric loading only. A value of 0 (or ALL) resets the default behavior for cyclically symmetric loading (where the loads are identical on all sectors).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MOVE

Specifies if the program should move high- or low-edge component nodes paired within the specified tolerance (TOLER) to create *precisely matching* pairs.

Value1

The flag value.

0

Do not move edge component nodes (default)

1 or HIGH

Move the high-edge component nodes to precisely match the low-edge component nodes

-1 or LOW

Move the low-edge component nodes to precisely match the high-edge component nodes

MSUP

This flag is used to limit the results written to the `Jobname.mode` and `Jobname.rst` files in a modal cyclic symmetry analysis. In a linear perturbation analysis, the modal analysis and the first load step of the preceding base analysis must be set to the same value.

Value1

The flag value.

0 (OFF or NO)

Write results for the base and duplicate sectors to the `Jobname.mode` and `Jobname.rst` files.

1 (ON or YES)

Write only the base sector results to the `Jobname.mode` and `Jobname.rst` files for use in a subsequent mode-superposition-based analysis. Default, except for cyclic unsymmetric modal, LANPCG, and SNODE solutions, which use *Value1* = 0 as the default. This option is not valid for cyclic unsymmetric modal, LANPCG, and SNODE solutions.

STATUS

List the solution option settings active for the cyclic model.

TOLER

The tolerance used to determine whether a node on the low edge is paired with a node on the high edge.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value1

The tolerance value.

Greater than 0

The absolute distance tolerance for automatic sector-boundary detection and low-/high-edge component node pairing

Less than 0

The relative tolerance for automatic sector-boundary detection and low-/high-edge component node pairing. In this case, the tolerance is *Value1* * *Length*, where *Length* is the length of the diagonal of an imaginary box enclosing the model

0

Tolerance is set to -1.0 x 10^{-4} (default)

Value2

ANGTOL = Maximum allowable angle tolerance. (default = 0.01°)

The valid range for *ANGTOL* is model dependent.

If you input both the number of sectors and a sector angle, the angle must match 360/(number of sectors) within *ANGTOL*.

If you input only a sector angle, it must divide evenly into 360° within *ANGTOL*.

If you input a sector angle, the final cyclic sector must span that angle within *ANGTOL*.

For auto detected sector angle, the final cyclic sector must span 360/(number of sectors) within *ANGTOL*, everywhere along the LOW/HIGH boundaries.

If *ANGTOL* is too small, your CAD or FEA model may not be accurate enough to allow auto detection or verification.

If *ANGTOL* is too large, you may get an unexpected or incorrect boundary definition, or in other cases fail to detect the boundaries.

For some difficult cases from FEA models (not solid models), you may need to change the value of *FACETOL* to achieve auto detection. Please refer to **CYCOPT** Auto Detection Tolerance Adjustments for Difficult Cases for more information about auto-detection and the **CYCOPT (p. 466)** command.

USRROT

Flag specifying whether the program should override automatic nodal rotations to edge components and allow you to apply nodal rotations manually.

Value1

The flag value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 (OFF or NO)

Allow automatic node rotation (default)

1 (ON or YES)

Suppress automatic node rotation. If you select this option, you *must* apply appropriate nodal rotations to all edge component nodes; otherwise, your analysis will yield incorrect solution results.

LOW

Suppresses automatic rotation of low-edge component nodes only, allowing you to apply them manually. Automatic rotation of high-edge component nodes occurs to produce the matching edge nodes required for a valid cyclic solution.

HIGH

Suppresses automatic rotation of high-edge component nodes only, allowing you to apply them manually. Automatic rotation of low-edge component nodes occurs to produce the matching edge nodes required for a valid cyclic solution.

Notes

The program solves a cyclically symmetric model (set up via the **CYCLIC (p. 462)** command during preprocessing) at the harmonic indices specified via the **CYCOPT (p. 466)** command.

The **CYCOPT (p. 466)**,COMBINE option is an alternative to the **/CYCEXPAND (p. 454)** command and is especially useful for testing purposes. However, Ansys, Inc. recommends specifying COMBINE only when the number of sectors is relatively small. (The option expands nodes and elements into the full 360° and can slow postprocessing significantly.

If you issue a **CYCOPT (p. 466)**,TOLER command to set a tolerance for edge-component pairing *before* issuing the **CYCLIC (p. 462)** command, the **CYCLIC (p. 462)** command uses the specified tolerance when performing automatic edge-component detection.

In cases involving non-cyclically symmetric loading (that is, when LDSECT > 0), the underlying command operations create or modify the required SECTOR tabular boundary condition (BC) data to apply on the appropriate sector. Therefore, it is not necessary to manipulate tables for situations where the applied BC is not a function of other tabular BC variables such as TIME, X, Y, Z, and so on.

To delete a previously applied load on a specified sector, issue an **FDELE (p. 742)** command.

Because edge nodes are rotated into the cyclic coordinate system during solution, any applied displacements or forces on sector edges will be in the cyclic coordinate system.

The **CYCOPT (p. 466)** command is valid in the preprocessing and solution stages of an analysis.

To learn more about analyzing a cyclically symmetric structure, see the *Cyclic Symmetry Analysis Guide*.

Distributed-Memory Parallel (DMP) Restriction

The COMBINE option is not supported in a DMP solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Cyclic Sector>Cyclic Options

Main Menu>Solution>Solve>Cyclic Options

CYCPHASE, *TYPE*, *OPTION*

Provides tools for determining minimum and maximum possible result values from frequency couplets produced in a modal cyclic symmetry analysis.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TYPE

The type of operation requested:

DISP

Calculate the maximum and minimum possible displacement at each node in the original sector model. Store the values and the phase angle at which they occurred.

STRESS

Calculate the maximum and minimum possible stresses at each node in the original sector model. Store the values and the phase angle at which they occurred.

STRAIN

Calculate the maximum and minimum possible strains at each node in the original sector model. Store the values and the phase angle at which they occurred.

ALL

Calculate the maximum and minimum possible displacement, stress and strain at each node in the original sector model. Store the values and the phase angle at which they occurred.

GET

Places the value of a MAX or MIN item into the _CYCVALUE parameter, the node for that value in the _CYCNODE parameter, and the phase angle for the value in the _CYCPHASE parameter.

PUT

Put resulting sweep values for printing (via the **PRNSOL (p. 1505)** command) or plotting (via the **PLNSOL (p. 1425)** command).

LIST

List the current minimum/maximum displacement, stress and strain nodal values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STAT

Summarize the results of the last phase sweep.

CLEAR

Clear phase-sweep information from the database.

OPTION

If *TYPE* = DISP, STRAIN, STRESS or ALL, controls the sweep angle increment to use in the search:

Angle

The sweep angle increment in degrees, greater than 0.1 and less than 10. The default is 1.

If *TYPE* = PUT, controls which values are placed onto the model:

MAX

Put all existing nodal maximum values onto the model. This option is the default.

MIN

Put all existing nodal minimum values onto the model.

If *TYPE* = GET, controls the values placed into cyclic parameters:

Item

Specifies the type of values on which to operate:

U -- Displacement
S -- Stress
EPEL -- Strain

Comp

Specifies the specific component of displacement, stress or strain for which to get information:

X,Y,Z -- Basic components
XY,YZ,XZ -- Shear components
1,2,3 -- Principal values
EQV -- Equivalent value
SUM -- USUM

MxMn

Specifies whether the requested value information is for the maximum or minimum value:

MAX -- Maximum value.
MIN -- Minimum value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

No defaults are available for the **CYCPHASE (p. 472)** command. You must specify an argument (*TYPE*) when issuing the command. Other values which may be necessary (*OPTION*) depend upon which *TYPE* argument you specify.

Notes

When you expand the results of a modal cyclic symmetry analysis (via the **/CYCEXPAND (p. 454)** or **EXPAND (p. 711)** command), the program combines the real and imaginary results for a given nodal diameter, assuming no phase shift between them; however, the modal response can occur at *any* phase shift.

CYCPHASE (p. 472) response results are valid only for the first cyclic sector. To obtain the response at *any* part of the expanded model, Ansys, Inc. recommends using cyclic symmetry results expansion at the phase angle obtained via **CYCPHASE (p. 472)**.

The phase angles returned by **CYCPHASE (p. 472)** contain the minimum and maximum values for USUM, SEQV and other scalar principal stress and strain quantities; however, they do not always return the true minimum and maximum values for directional quantities like UX or SX unless the values fall in the first sector.

CYCPHASE (p. 472) does not consider midside node values when evaluating maximum and minimum values, which may affect display quantities but no others. (Typically, the program ignores midside node stresses and strains during postprocessing.)

Issuing **CYCPHASE (p. 472)**,PUT clears the result values for midside nodes on high order elements; therefore, this option sets element faceting (**/EFACET (p. 602)**) to 1. The command reports that midside nodal values are set to zero and indicates that element faceting is set to 1.

If the sweep values are available after issuing a **CYCPHASE (p. 472)**,PUT command, the **PRNSOL (p. 1505)** or **PLNSOL (p. 1425)** command will print or plot (respectively) the sweep values of structure displacement Ux, Uy, Uz, component stress/strain X, Y, Z, XY, YZ, ZX, principal stress/strain 1, 2, 3 and equivalent stress/strain EQV. The vector sum of displacement (USUM) and stress/strain intensity (SINT) are not valid phase-sweep results.

You can specify any coordinate system via the **RSYS (p. 1639)** command for displaying or printing **CYCPHASE (p. 472)** results. However, after **CYCPHASE (p. 472)** results have been extracted, you cannot then transform them via the **RSYS (p. 1639)** command. If you try to do so, the program issues a warning message.

The **CYCPHASE (p. 472)** command is valid in **/POST1 (p. 1461)** and for cyclically symmetric models only.

To learn more about analyzing a cyclically symmetric structure, see the *Cyclic Symmetry Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Clear

Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Get

Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>List

Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Phase Sweep

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Put Phase Results

Main Menu>General Postproc>Cyclic Analysis>Cyclic Phase>Status

CYCSPEC, *LABEL*, *Node*, *Item*, *Comp*

Defines the set of result items for a subsequent CYCCALC (p. 453) command in postprocessing a cyclic harmonic mode-superposition analysis.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LABEL

One of the following labels:

ADD

Adds a new specification to the set (default). The maximum number of specifications that can be defined is 50.

LIST

Lists the current set of specifications. *Node*, *Item*, *Comp* are ignored.

ERASE

Erases the current set of specifications. *Node*, *Item*, *Comp* are ignored.

DELETE

Deletes an existing specification. *Item*, *Comp* are ignored.

Node

The node at which to evaluate the results. If *Node* is a nodal component, then all nodes in the component are included. All sectors containing this node (or set of nodes) are evaluated.

For *LABEL* = DELETE, use *Node* to indicate which specification in the set to delete.

Item

Specifies the type of values to evaluate:

U

Displacement

S

Stress

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EPEL

Elastic strain

Comp

Specifies the specific component of displacement, stress, or strain to evaluate:

X,Y,Z

Direct components

XY,YZ,XZ

Shear components (stress and strain only)

1,2,3

Principal values (stress and strain only)

EQV

Equivalent value (stress and strain only)

SUM

Vector sum (displacement only)

NORM

L2 norm for the set of nodes (displacement only)

Command Default

No defaults are available for the **CYCSPEC (p. 475)** command. You must issue this command to define the set of result items for evaluation in a subsequent **CYCCALC (p. 453)** command used in computing results of a cyclic harmonic mode-superposition analysis.

Notes

Up to 50 specifications can be defined for use in a subsequent **CYCCALC (p. 453)** command. If more than 50 specifications are desired, erase the table after the **CYCCALC (p. 453)** operation and add new specifications and repeat the **CYCCALC (p. 453)** command. All the specified nodes, items, and components are evaluated for all sectors and the maximum amplitude value output. For combined stresses and strains (*Comp* = 1,2,3 or EQV) or displacement vector sum (*Comp* = SUM), a 360 degree phase sweep is performed at each location to determine the maximum.

Additional POST1 controls are used to refine the specification. For component values, components are in the **RSYS (p. 1639)** direction. For shell elements, the results are at the **SHELL (p. 1784)** location. For EPEL,EQV, the results are based on the *EFFNU* value on the **AVPRIN (p. 210)** command. The controls active when the **CYCCALC (p. 453)** command is issued determine the result values. If results at another **SHELL (p. 1784)** location are desired, issue the new **SHELL (p. 1784)** command and then re-issue the **CYCCALC (p. 453)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a single node is input, the *Item*/*Comp* value at that location in each sector is output. If a node component is given, then the maximum *Item*/*Comp* value within the set of nodes of each sector is output, one value for each sector (the node of the maximum may vary from sector to sector). For stress and strain items, only corner nodes are valid.

For the displacement norm option (*Item* = U, *Comp* = NORM), the L2 norm computed from all the nodes in the component is output, one per sector.

Menu Paths

This command cannot be accessed from a menu.

CYL4, *XCENTER*, *YCENTER*, *RAD1*, *THETA1*, *RAD2*, *THETA2*, *DEPTH*

Creates a circular area or cylindrical volume anywhere on the working plane.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCENTER, YCENTER

Working plane X and Y coordinates of the center of the circle or cylinder.

RAD1, RAD2

Inner and outer radii (either order) of the circle or cylinder. A value of zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid circle or cylinder.

THETA1, THETA2

Starting and ending angles (either order) of the circle or faces of the cylinder. Used for creating a partial annulus or partial cylinder. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°. See the *Modeling and Meshing Guide* for an illustration.

DEPTH

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cylinder. If *DEPTH* = 0 (default), a circular area is created on the working plane.

Notes

Defines a circular area anywhere on the working plane or a cylindrical volume with one face anywhere on the working plane. For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines) and they will be connected with two surface areas (each spanning 180°). See the **CYL5 (p. 478)**, **PCIRC (p. 1369)**, and **CYLIND (p. 479)** commands for alternate ways to create circles and cylinders.

When working with a model imported from an IGES file (DEFAULT import option), you must provide a value for *DEPTH* or the command will be ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Annulus

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Partial Annulus

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Solid Circle

Main Menu>Preprocessor>Modeling>Create>Primitives>Solid Cylindr

Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Hollow Cylinder

Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Partial Cylinder

Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>Solid Cylinder

---

CYL5, *XEDGE1*, *YEDGE1*, *XEDGE2*, *YEDGE2*, *DEPTH*

Creates a circular area or cylindrical volume by end points.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XEDGE1, YEDGE1

Working plane X and Y coordinates of one end of the circle or cylinder face.

XEDGE2, YEDGE2

Working plane X and Y coordinates of the other end of the circle or cylinder face.

DEPTH

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the cylinder. If *DEPTH* = 0 (default), a circular area is created on the working plane.

Notes

Defines a circular area anywhere on the working plane or a cylindrical volume with one face anywhere on the working plane by specifying diameter end points. For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines) and they will be connected with two surface areas (each spanning 180°). See the **CYL4 (p. 477)**, **PCIRC (p. 1369)**, and **CYLIND (p. 479)** commands for alternate ways to create circles and cylinders.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>By End Points

Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>By End Pts & Z

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CYLIND, *RAD1*, *RAD2*, *Z1*, *Z2*, *THETA1*, *THETA2*

Creates a cylindrical volume centered about the working plane origin.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RAD1, RAD2

Inner and outer radii (either order) of the cylinder. A value of zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid cylinder.

Z1, Z2

Working plane Z coordinates of the cylinder. If either *Z1* or *Z2* is zero, one of the faces of the cylinder will be coplanar with the working plane.

THETA1, THETA2

Starting and ending angles (either order) of the cylinder. Used for creating a cylindrical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the *Modeling and Meshing Guide* for an illustration.

Notes

Defines a cylindrical volume centered about the working plane origin. The top and bottom faces are parallel to the working plane but neither face need be coplanar with (that is, "on") the working plane. The cylinder must have a spatial volume greater than zero. (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) For a solid cylinder of 360°, the top and bottom faces will be circular (each area defined with four lines), and they will be connected with two areas (each spanning 180°.) See the **CYL4 (p. 477)** and **CYL5 (p. 478)** commands for alternate ways to create cylinders.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Cylinder>By Dimensions

CZDEL, *grp1*, *grp2*, *grp3*

Edits or clears cohesive zone sections.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

grp1

Initial group of cohesive zone elements to be deleted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

grp2

Final group of cohesive zone elements to be deleted.

grp3

Increment for selected groups.

Notes

The **CZDEL (p. 479)** command edits or deletes the interface elements and nodes, along with the associated changes made to the underlying plane or solid elements created during a previous **CZMESH (p. 480)** operation.

Each **CZMESH (p. 480)** operation will create groups of elements and nodes with component names in the format CZME_EL01 (elements) and CZME_ND01 (nodes). The final number of this format will be the number used for *grp1* and *grp2*. If *grp1* = ALL, all nodes and elements created by the **CZMESH (p. 480)** command will be deleted. After using **CZDEL (p. 479)**, all the user-defined components will be unselected.

The **CZDEL (p. 479)** command is valid for structural analyses only.

Menu Paths

This command cannot be accessed from a menu.

CZMESH,*ecomps1*,*ecomps2*,*KCN*,*KDIR*,*VALUE*,*CZTOL*

Create and mesh an interface area composed of cohesive zone elements.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ecomps1

Component name or number for the group of plane or solid structural elements adjacent to the interface being meshed.

ecomps2

Component name or number for the opposing (from *ecomps1*) group of plane or solid structural elements adjacent to the interface being meshed.

KCN

Coordinate system number for the separation surface and normal direction. (if *ecomps1* and *ecomps2* not specified)

KDIR

Direction (x, y, or z) normal to separation surface in the *KCN* coordinate system (if *ecomps1* and *ecomps2* not specified).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VALUE

Coordinate value along the *KDIR* axis at which to locate the interface (if *ecomps1* and *ecomps2* not specified).

CZTOL

Optional absolute tolerance about *VALUE* (if *ecomps1* and *ecomps2* not specified). Allows nodes occurring slightly above or below the separation to be grouped properly. The following expression represents the default value:

$$\frac{\sqrt{\Delta X^2+\Delta Y^2+\Delta Z^2}}{1000}$$

Where ΔX, ΔY, and ΔZ are the dimensions of the model based on nodal locations (that is, ΔX = X*max* - X*min*).

Notes

CZMESH (p. 480) is used on a mesh with shared nodes at the interface.

If *ecomps1* and *ecomps2* are specified, the **CZMESH (p. 480)** command creates/meshes interface elements (INTER202, INTER203, INTER204, INTER205) along the boundary between the two components or groups of elements.

The elements in each of the components or groups of elements share nodes with each other and also with the interface elements. This one-element thick boundary of interface elements splits the body between the two components or groups of elements.

Subsequent separation (delamination and failure) of the interface zone results in an increasing displacement between the nodes (within the interface element) along the cohesive zone elements. Unless otherwise specified, the **CZMESH (p. 480)** command analyzes the configuration and geometry of the adjacent structural elements and provides the appropriate interface element.

The **CZMESH (p. 480)** operation copies any nodal temperatures that you have defined on the split surface of the original mesh from the original nodes to the newly created coincident duplicate nodes. Displacements, forces, and other boundary conditions, however, are not copied; therefore, apply boundary conditions and loadings after issuing **CZMESH (p. 480)**.

If using **CZMESH (p. 480)** to generate interface elements (INTER202 and INTER205) in a VCCT-based crack-growth simulation, be aware that those elements do not support degenerate shapes. Examine the resulting mesh, therefore, to verify correct element connectivity around the crack tip.

This command does not support mesh regions with non-solid elements (such as surface-effect and shell elements).

This command is valid for structural analyses only.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

D Commands

D, *Node*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *MESHFLAG*

Defines degree-of-freedom constraints at nodes.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Node

Node at which constraint is to be specified. If ALL, *NEND* and *NINC* are ignored and constraints are applied to all selected nodes (**NSEL (p. 1266)**). If *Node* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *Node*.

Lab

Valid degree-of-freedom label. If ALL, use all appropriate labels.

Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping).

Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature).

Electric labels: VOLT (voltage); EMF (electromotive force).

Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential).

Acoustic labels: PRES (pressure); UX, UY, or UZ (displacements for FSI coupled elements); ENKE (acoustic energy density); VX, VY or VZ (velocity); TEMP (temperature).

Pore fluid labels: PRES (pore pressure); UX, UY, or UZ (displacements); TEMP (temperature).

Diffusion labels: CONC (concentration).

For structural static and transient analyses, translational and rotational velocities are also valid loads. Use these labels: VELX, VELY, VELZ (translational velocities); OMGX, OMGY, OMGZ (rotational velocities).

For structural analyses, HDSP (hydrostatic pressure) is also valid. However, HDSP is *not* included when *Lab* = ALL.

For structural transient analyses, the following acceleration loads are also valid: ACCX, ACCY, ACCZ (translational accelerations); DMGX, DMGY, DMGZ (rotational accelerations). The velocity and acceleration loads are *not* included when *Lab* = ALL.

If the node is connected to an ELBOW290 element, the following pipe cross-section degree-of-freedom labels are also valid: SE, SO, SW, SRA, and SRT. (For details, see the ELBOW290 documenta-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tion.) The degrees of freedom are *not* included when *Lab* = ALL. To constrain *all* cross-section degrees of freedom, specify *Lab* = SECT.

The PRES degree of freedom is also available for porous media problems.

Value

Degree-of-freedom value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent (%) signs (for example, **D (p. 483)**,*Node*,TEMP,%*tabname*%). To define a table, issue ***DIM (p. 530)**.

If *Value* = SUPPORT, you can specify pseudo-constraints when using residual vectors in a modal analysis (**RESVEC (p. 1603)**,ON) or CMS analysis (**CMSOPT (p. 378)**,RFFB).

If the enforced motion is active in the modal analysis (**MODCONT (p. 1132)**,,on), *Value* is the base identification number. It should be an integer greater than or equal to 1 and less than 10000.

VALUE2

Second degree-of-freedom value (if any). If the analysis type and the degree of freedom allow a complex input, *Value* (above) is the real component and *VALUE2* is the imaginary component.

NEND, NINC

Specifies the same values of constraint at the range of nodes from *Node* to *NEND* (defaults to *Node*), in steps of *NINC* (defaults to 1).

Lab2, Lab3, Lab4, Lab5, Lab6

Additional degree-of-freedom labels. The same values are applied to the nodes for these labels.

MESHFLAG

Specifies how to apply constraint on the mesh. Valid in a nonlinear adaptivity analysis when *Lab* = UX / UY / UZ and *Node* is not a component name.

0 – Constraint applied on the current mesh (default).

1 – Constraint applied on the initial mesh for nonlinear adaptivity. (*NEND* and *NINC* are not valid.)

Notes

The available degrees of freedom per node are listed under **Degrees of Freedom** in the input table for each element type in the *Element Reference*. Degrees of freedom are defined in the nodal coordinate system. The positive directions of structural translations and rotations are along and about the positive nodal axes directions. Structural rotations should be input in radians. The node and the degree-of-freedom label must be selected (**NSEL (p. 1266)**, **DOFSEL (p. 569)**).

In a structural analysis, you can apply only one displacement, velocity, or acceleration load at any degree of freedom. If multiple loads are specified, the last applied load overrides the previous ones. For example, the following commands apply loads to node 100:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

D,100,UX,*Value*

D,100,VELX,*Value*

In this case, the velocity load (VELX) applied in the last command will override the displacement load (UX).

For elements used in static and low frequency electromagnetic analysis (SOLID236 and SOLID237), the AZ degree of freedom is not a z-component of a vector potential, but rather the flux contribution on the element edge. To specify a flux-parallel condition, set AZ = 0. For more information, see 3D Magnetostatics and Fundamentals of Edge-based Analysis in the *Low-Frequency Electromagnetic Analysis Guide*.

For ELBOW290 cross-section degrees of freedom (*Lab* = SE, SO, SW, SRA, SRT, or SECT), the **D (p. 483)** command can only specify fixed constraints. The degree-of-freedom value must be zero; no other values are valid.

For hydrostatic fluid elements (HSFLD241 and HSFLD242), the HDSP degree-of-freedom constraint at the pressure node prescribes the pressure value for all the fluid elements sharing the pressure node.

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree-of-freedom labels: Electric (VOLT), structural (UX, UY, UZ, ROTX, ROTY, ROTZ, and velocity and acceleration loads VELX, VELY, VELZ, OMGX, OMGY, OMGZ, ACCX, ACCY, ACCZ, DMGX, DMGY, DMGZ), acoustic (PRES, UX, UY, UZ, ENKE), temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP), diffusion (CONC). All labels are valid only in static (**ANTYPE (p. 162)**,STATIC) and full transient (**ANTYPE (p. 162)**,TRANS) analyses.

In a mode-superposition harmonic or transient analysis, you must apply the constraints in the modal portion of the analysis for residual vector (Using the Residual Vector or the Residual Response Method to Improve Accuracy) and enforced motion (Enforced Motion Method for Mode-Superposition Transient and Harmonic Analyses) definitions.

%_FIX% is a Mechanical APDL reserved table name. When *VALUE* is set to %_FIX%, the program prescribes the degree of freedom to the current relative displacement value. This option is valid for the following labels: UX, UY, UZ, ROTX, ROTY, ROTZ. Alternatively, functions UX(), UY(), etc. can be used. (See ***GET (p. 797)** for a list of available functions.) In most cases, %_FIX% usage is efficient and recommended for all structural degrees of freedom.

When *Value* = SUPPORT, specify only the minimum number of displacement constraints necessary to prevent rigid body motion: three constraints (or fewer, depending on the element type) for 2D models and six (or fewer) for 3D models.

If constraints and initial conditions (**IC (p. 921)**) are applied at the same node, the constraint specification overrides. This combination is useful when a constraint degree-of-freedom value needs to start with a nonzero value at time = 0.0. For example, if the constraint degree-of-freedom value is prescribed to be a cosine function, then specifying an initial condition for the same node and degree of freedom ensures that the initial value for the constraint degree of freedom at time = 0.0 is same as the cosine function evaluated at time = 0.0. If initial conditions are not specified, the constraint degree-of-freedom value ramps from zero in the first substep of the first load step.

If more than one rotational degrees of freedom are constrained with non-zero rotations (ROTX, ROTY, ROTZ), rotational velocities (OMGX, OMGY, OMGZ), or rotational accelerations (DMGX, DMGY, DMGZ), then the rotation of the constrained node from its initial configuration to its final configuration depends

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on the combination and the sequence in which the constraints are applied. See Rotations in a Large-Deflection Analysis in *Structural Analysis Guide*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Node Components

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>BasePSD>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>BasePSD>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Node Components

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>BasePSD>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Nodes

Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>BasePSD>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Nodes

---

DA,*AREA*,*Lab*,*Value1*,*Value2*

Defines degree-of-freedom constraints on areas.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Area on which constraints are to be specified. If ALL, apply to all selected areas (**ASEL (p. 192)**). If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

Lab

Symmetry label (see 1 below):

SYMM

Generate symmetry constraints. Requires no *Value1* or *Value2*.

ASYM

Generate antisymmetry constraints. Requires no *Value1* or *Value2*.

Mechanical APDL degree-of-freedom labels:

UX

Displacement in X direction.

UY

Displacement in Y direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UZ

Displacement in Z direction.

ROTX

Rotation about X axis.

ROTY

Rotation about Y axis.

ROTZ

Rotation about Z axis.

HDSP

Hydrostatic pressure.

PRES

Pressure.

TEMP, TBOT, TE2, TE3, . . ., TTOP

Temperature.

MAG

Magnetic scalar potential (see 2 below).

VOLT

Electric scalar potential (see 3 below).

AZ

Magnetic vector potential in Z direction (see 4 below).

CONC

Concentration.

ALL

Applies all appropriate degree-of-freedom labels *except* HDSP.

Footnotes for *Lab*:

1. Symmetry and antisymmetry constraints are generated as described for the **DSYM (p. 580)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If *Lab* = MAG and *Value1* = 0, this sets the flux-normal condition for the magnetic scalar potential formulations (MSP) (A flux-parallel condition is the natural boundary condition for MSP.)
3. If *Lab* = VOLT and *Value1* = 0, the J-normal condition is set (current density (J) flow normal to the area). (A J-parallel condition is the natural boundary condition.)
4. For elements SOLID236 and SOLID237, if *Lab* = AZ and *Value1* = 0, this sets the flux-parallel condition for the edge formulation. (A flux-normal condition is the natural boundary condition.) Do *not* use the **DA (p. 487)** command to set the edge-flux degree of freedom, AZ to a nonzero value.

Value1

Value of degree of freedom or table name reference on the area. Valid for all degree-of-freedom labels. To specify a table, enclose the table name in % signs (for example, **DA (p. 487)**,*AREA*,TEMP,%*tabname*%). Use the ***DIM (p. 530)** command to define a table.

Value2

For MAG and VOLT degrees of freedom:

Value of the imaginary component of the degree of freedom.

Notes

You can transfer constraints from areas to nodes via **DTRAN (p. 583)** or **SBCTRAN (p. 1647)**. See **DK (p. 541)** for information about generating other constraints on areas.

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree-of-freedom labels: Electric (VOLT), Structural (UX, UY, UZ, ROTX, ROTY, ROTZ), Acoustic (PRES, UX, UY, UZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).

Constraints specified by the **DA (p. 487)** command can conflict with other specified constraints. See Resolution of Conflicting Constraint Specifications\ in the *Basic Analysis Guide* for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>J-Normal>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>EdgeMVP>Flux Par'l>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>Flux Normal>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DADELE, *AREA*, *Lab*

Deletes degree-of-freedom constraints on an area.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Area for which constraints are to be deleted. If ALL, delete for all selected areas (**ASEL (p. 192)**). If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You can substitute a component name for *AREA*.

Lab

Valid constraint labels are:

ALL

All constraints.

SYMM

Symmetry constraints.

ASYM

Antisymmetry constraints.

UX

Displacement in X direction.

UY

Displacement in Y direction.

UZ

Displacement in Z direction.

ROTX

Rotation about X axis.

ROTY

Rotation about Y axis.

ROTZ

Rotation about Z axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRES

Pressure.

TEMP, TBOT, TE2, TE3, . . ., TTOP

Temperature.

MAG

Magnetic scalar potential.

VOLT

Electric scalar potential.

AZ

Magnetic vector potential in Z direction (see notes).

CONC

Concentration.

Notes

Deletes the degree of freedom constraints at an area (and all corresponding finite element constraints) previously specified with the **DA (p. 487)** command. See the **DDELE (p. 505)** command for delete details.

If the multiple species labels have been changed to user-defined labels via the MSSPEC command, use the user-defined labels.

See the **DA (p. 487)** or the **DA (p. 487)** commands for details on element applicability.

> **Warning:**
>
> *On previously meshed areas*, **all** constraints on affected nodes will be deleted, whether or not they were specified by the **DA (p. 487)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Areas

Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Areas

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Areas

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Areas

Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Areas

Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Areas

---

DALIST,*AREA*

Lists the DOF constraints on an area.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

List constraints for this area. If ALL (default), list for all selected areas (**ASEL (p. 192)**). If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *AREA*.

Notes

Lists the degree of freedom constraints on an area previously specified with the **DA (p. 487)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Areas

Utility Menu>List>Loads>DOF Constraints>On Picked Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DAMORPH,*AREA*,*XLINE*,*RMSHKY*

Move nodes in selected areas to conform to structural displacements.

PREP7 (p. 22): Morphing (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Non-structural area to which mesh movement (morph) applies. If ALL, apply morphing to all selected areas [ASEL]. If *AREA* = P, graphical picking is enabled. A component may be substituted for *AREA*.

XLINE

Lines to be excluded from morphing. If ALL, exclude all selected lines [LSEL] from morphing. If *XLINE* = P, graphical picking is enabled. A component may be substituted for *XLINE*. If *XLINE* is blank (default), allow morphing of nodes attached to lines of the selected areas (*AREA*) which are not shared by unselected areas. See Notes for clarification.

RMSHKY

Remesh flag option:

0

Remesh the selected non-structural areas only if mesh morphing fails.

1

Remesh the selected non-structural areas and bypass mesh morphing.

2

Perform mesh morphing only and do not remesh.

Notes

The selected areas should include only non-structural regions adjacent to structural regions. **DAMORPH (p. 494)** will morph the non-structural areas to coincide with the deflections of the structural regions.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing **DAMORPH (p. 494)**.

By default, nodes attached to lines can move along the lines, or off the lines (if a line is interior to the selected areas). You can use *XLINE* to restrain nodes on certain lines.

By default (*RMSHKEY* = 0), **DAMORPH (p. 494)** will remesh the selected non-structural areas entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the **DAMORPH (p. 494)** command requires that these be removed before remeshing can take place.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exercise care with initial conditions defined by the **IC (p. 921)** command. Before a structural analysis is performed for a sequentially coupled analysis, the **DAMORPH (p. 494)** command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE (p. 923)** to delete the initial conditions.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Areas

---

DAMPOPT, *Option*, *Value*

Sets damped eigensolver options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Damped eigensolver option:

SHIFT

Activate or deactivate the shift strategy of the damped eigensolver to move to a frequency range of interest. The *FREQB* value of the **MODOPT (p. 1140)** command is used to choose the first frequency shift (see notes for details).

Valid input for *Value* when *Option* = SHIFT:

OFF

Do not activate the shift strategy.

ON

Activate the shift strategy (default).

MEMORY

Controls the memory allocation strategy for the Damped eigensolver.

Valid input for *Value* when *Option* = MEMORY:

DEFAULT

Default memory configuration (default). Everything is determined dynamically with respect to current machine resources.

INCORE

Fully in-core memory configuration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MIX1

First level of mixed in-core / out-of-core configuration.

MIX2

Second level of mixed in-core / out-of-core configuration.

OUTOFCORE

Fully out-of-core memory configuration.

Value

Assigned value for the specified *Option* (as described above).

Command Default

By default, the shift strategy is not activated for the damped eigensolver. This ensures a faster solve process but limits the number of eigenvalues that can be found.

The default memory allocation strategy is used. Mechanical APDL evaluates the resources of the machine to choose the in-core or out-of-core mode.

Notes

DAMPOPT (p. 495) specifies options to be used with the damped eigensolver (**MODOPT (p. 1140)**,DAMP) during a modal analysis (**ANTYPE (p. 162)**,MODAL).

Activating the shift strategy (*Option* = SHIFT and *Value* = ON) enables the eigensolver to find higher frequency eigenvalues that might otherwise be missed. The SHIFT option has two objectives:

- Extract high frequency eigenvalues according to the *FREQB* argument specified with **MODOPT (p. 1140)**.
- Unlock an auto-shift feature, so the algorithm will chain several analyses automatically then aggregate the solutions in one single results set.

If *FREQE* is specified on **MODOPT (p. 1140)**, the specified value is used to filter the complex eigenfrequencies based on magnitude.

When the shift strategy is activated (**DAMPOPT (p. 495)**,SHIFT,ON)

- and *FREQB* on **MODOPT (p. 1140)** is specified, the damped eigensolver only produces eigenfrequencies with a real part greater than *FREQB*
- and *FREQB* on **MODOPT (p. 1140)** is not specified, the damped eigensolver finds eigenvalues closest to the 0 Hz point.

Memory Allocation Option (*Option* = MEMORY)

The damped eigensolver algorithm allocates two main pools of memory:

1. Memory for the internal damped eigensolver iterations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Memory for the specific damped eigensolver working arrays.

The following table shows how memory is allocated for each option.

Value **(*Option* = MEMORY)**	**Damped Eigensolver**	**Working Arrays**
INCORE	In-Core	In-Core
MIX1	In-Core	Out-of-Core
MIX2	Out-of-Core	In-Core
OUTOFCORE	Out-of-Core	Out-of-Core

The MIX1 configuration typically uses more memory than the MIX2 configuration, except when a large number of modes are requested for a small model.

Example Usage

Example: Shift Option Usage on **DAMPOPT** in the *Structural Analysis Guide*

Menu Paths

This command cannot be accessed from a menu.

DATA, *IR*, *LSTRT*, *LSTOP*, *LINC*, *Name*, *KCPLX*

Reads data records from a file into a variable.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

LSTRT

Start at location *LSTRT* (defaults to 1).

LSTOP

Stop at location *LSTOP* (defaults to *LSTRT*). Maximum location available is determined from data previously stored.

LINC

Fill every *LINC* location between *LSTRT* and *LSTOP* (defaults to 1).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

Eight character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

KCPLX

Complex number key:

0

Data stored as the real part of the complex number.

1

Data stored as the imaginary part of the complex number.

Notes

This command must be followed by a format statement (on the next line) and the subsequent data records, and all must be on the same file (that may then be read with the **/INPUT (p. 948)** command). The format specifies the number of fields to be read per record, the field width, and the placement of the decimal point (if one is not included in the data value). The read operation follows the available FORTRAN FORMAT conventions of the system. See the system FORTRAN manual for details. Any standard FORTRAN real format (such as (4F6.0), (F2.0,2X,F12.0), etc.) may be used. Integer (I), character (A), and list-directed (*) descriptors may *not* be used. *The parentheses must be included in the format*. Up to 80 columns per record may be read. Locations may be filled within a range. Previous data in the range will be overwritten.

Menu Paths

This command cannot be accessed from a menu.

DATADEF

Specifies "Directly defined data status" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>List>Status>General Postproc>Modify Results

/DBDECRYPT, *KeyA*, *KeyB*, *DataType*, *NUM1*, *NUM2*, *INC*

Controls decryption of material data in the database file.

APDL (p. 19): Encryption/Decryption (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KeyA

Decryption key A (32-character maximum). This key is used to decrypt the data in a one-level encryption or to control access to the data in a two-level encryption. Leave this field blank if you do not have key A.

KeyB

Decryption key B (32-character maximum). This key is used to decrypt the data in a two-level encryption. Leave this field blank if the database file is encrypted with one-level encryption.

DataType

Type of data to decrypt. Must be set to MAT for material data.

NUM1*, *NUM2*, *INC

Decrypt materials from material number *NUM1* to *NUM2* (defaults to *NUM1*) in steps of *INC* (defaults to 1). If *NUM1* = ALL (default), *NUM2* and *INC* are ignored.

Notes

This command decrypts data in the database file. It must be issued before resuming the database file (**RESUME (p. 1601)** command). Only *KeyA* is required for a one-level encryption. For a two-level encryption, inputting *KeyB* gives you partial access to the data. Inputting both *KeyA* and *KeyB* gives you full access.

For more information about using **/DBDECRYPT (p. 499)** in the encryption/decryption procedure, see Encrypting Material Data in the *Ansys Parametric Design Language Guide*. See also the **/DBENCRYPT (p. 500)** command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/DBENCRYPT, *KeyA*, *KeyB*, *DataType*, *NUM1*, *NUM2*, *INC*

Controls encryption of material data in the database file.

APDL (p. 19): Encryption/Decryption (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KeyA

Encryption key A (32-character maximum). This key is used to encrypt the data in a one-level encryption or to control access to the data in a two-level encryption.

KeyB

Encryption key B (32-character maximum). This key is used to encrypt the data in a two-level encryption . If *KeyB* is not specified, a one-level encryption is used to encrypt the data.

DataType

Type of data to encrypt. Must be set to MAT for material data.

NUM1*, *NUM2*, *INC

Encrypt materials from material number *NUM1* to *NUM2* (defaults to *NUM1*) in steps of *INC* (defaults to 1). If *NUM1* = ALL (default), *NUM2* and *INC* are ignored.

Notes

This command encrypts data in the database file. It must be issued before saving the database file (**SAVE (p. 1645)** command).

For a one-level encryption, specify only *KeyA* and set *NUM1* to ALL. (*NUM2* and *INC* are not used.)

For a two-level encryption, specify both *KeyA* and *KeyB*. Also specify *NUM1*, *NUM2*, and *INC* as needed.

For more information about using **/DBENCRYPT (p. 500)** in the encryption/decryption procedure, see Encrypting Material Data in the *Ansys Parametric Design Language Guide*. See also the **/DBDECRYPT (p. 499)** command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DCGOMG, *DCGOX*, *DCGOY*, *DCGOZ*

Specifies the rotational acceleration of the global origin.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DCGOX, DCGOY, DCGOZ

Rotational acceleration of the global origin about the acceleration system X, Y, and Z axes.

Notes

Specifies the rotational acceleration of the global origin about each of the acceleration coordinate system axes (**CGLOC (p. 319)**). Rotational accelerations may be defined in these analysis types:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC) -- full, VT [1], Krylov [1], or mode-superposition [2]
- Transient (**ANTYPE (p. 162)**,TRANS) -- full or mode-superposition [2]
- Substructuring (**ANTYPE (p. 162)**,SUBSTR)
- Modal (**ANTYPE (p. 162)**,MODAL)

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

See Acceleration Effect in the *Mechanical APDL Theory Reference* for details. Units are radians/time^2.

The **DCGOMG (p. 501)** command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *DCGOMG_X*, *DCGOMG_Y*, and *DCGOMG_Z* input values (***DIM (p. 530)**) for full transient and harmonic analyses.

Related commands are **ACEL (p. 92)**, **CGLOC (p. 319)**, **CGOMGA (p. 320)**, **DOMEGA (p. 571)**, and **OMEGA (p. 1324)**.

See Analysis Tools in the *Theory Reference* for more information.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Coriolis Effects

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Coriolis Effects

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Coriolis Effects

Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Coriolis Effects

---

DCUM, *Oper*, *RFACT*, *IFACT*, *TBASE*

Specifies that DOF constraint values are to be accumulated.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Accumulation key:

REPL

Subsequent values replace the previous values (default).

ADD

Subsequent values are added to the previous values.

IGNO

Subsequent values are ignored.

RFACT

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

TBASE

Base temperature for temperature difference. Used only with temperature degree of freedom. Scale factor is applied to the temperature difference (*T*-*TBASE*) and then added to *TBASE*. *T* is the current temperature.

Command Default

Replace previous values.

Notes

Allows repeated degree of freedom constraint values (displacement, temperature, etc.) to be replaced, added, or ignored. Operations apply to the selected nodes (**NSEL (p. 1266)**) and the selected degree of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

freedom labels (**DOFSEL (p. 569)**). This command also operates on velocity and acceleration loads applied in a structural analysis.

The operations occur when the next degree of freedom constraints are defined. For example, issuing the command **D (p. 483)**,1,UX,.025 after a previous **D (p. 483)**,1,UX,.020 causes the new value of the displacement on node 1 in the x-direction to be 0.045 with the add operation, 0.025 with the replace operation, or 0.020 with the ignore operation. Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a displacement of 0.070. Scale factors are applied even if no previous values exist. Issue **DCUM (p. 502)**,STAT to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note:

FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

DCUM (p. 502) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Constraints

Main Menu>Solution>Define Loads>Settings>Replace vs Add>Constraints

---

DDASPEC,*KeyRef*,*Shptyp*,*MountLoc*,*Deftyp*,*Amin*

Specifies the shock spectrum computation constants for DDAM analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KeyRef

Key for reference catalog:

1

The spectrum computation constants are based on NRL-1396 (default). For more information, see Dynamic Design Analysis Method in the *Mechanical APDL Theory Reference*

Shptyp

Select the ship type:

SUBM

Submarine

SURF

Surface ship

MountLoc

Select the mounting location:

HULL

Hull mounting location. These structures are mounted directly to basic hull structures like frames, structural bulkheads below the water line, and shell plating above the water line.

DECK

Deck mounting location. These structures are mounted directly to decks, non-structural bulkheads, or to structural bulkheads above the water line.

SHEL

Shell plating mounting location. These structures are mounted directly to shell plating below the water line without intervening foundations.

Deftyp

Select the deformation type:

ELAS

Elastic deformation (default)

PLAS

Elastic-plastic deformation

Amin

Minimum acceleration value. It defaults to 6g, where g is the acceleration due to gravity.

Notes

The excitation along one of the fore and aft, vertical or athwartship directions is required to calculate the spectrum coefficients. Issue the **SED (p. 1704)** command before issuing **DDASPEC (p. 503)**. For example, if you want to apply the excitation along the fore and aft direction, you should specify *SEDX* = 1.0 on **SED (p. 1704)**. Similarly, for excitation along vertical or athwartship direction, specify *SEDY* = 1.0 or *SEDZ* = 1.0, respectively, on **SED (p. 1704)**.

ADDAM (p. 96) and **VDDAM (p. 2081)** may alternatively be used to calculate spectrum coefficients.

In order to perform a DDAM spectrum analysis using a units system other than BIN (default), you must specify the units system complying with the mass and length units of the model using the **/UNITS (p. 2054)** command. Issue the **/UNITS (p. 2054)** command before defining the shock spectrum computation constants (**DDASPEC (p. 503)**). The DDASPEC command is not supported with the user-defined units system (`Label` = USER on **/UNITS (p. 2054)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

DDELE, *NODE*, *Lab*, *NEND*, *NINC*, *Rkey*, *IRSTRTFLAG*

Deletes degree-of-freedom constraints.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node for which constraint is to be deleted. If ALL, *NEND* and *NINC* are ignored and constraints for all selected nodes (**NSEL (p. 1266)**) are deleted. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab

Valid degree of freedom label. If ALL, use all selected labels (**DOFSEL (p. 569)**). Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Acoustic labels: PRES (pressure); UX, UY, or UZ (displacements for FSI coupled elements). Electric label: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential). Diffusion label: CONC (concentration).

In structural analyses, the following velocity and acceleration load labels are also valid: VELX, VELY, VELZ (translational velocities); OMGX, OMGY, OMGZ (rotational velocities); ACCX, ACCY, ACCZ (translational accelerations); DMGX, DMGY, DMGZ (rotational accelerations).

In structural analyses, HDSP (hydrostatic pressure) is also valid.

If the node is connected to an ELBOW290 element, the following pipe cross-section degree of freedom labels are also valid: SE, SO, SW, SRA, and SRT. (For details, see the ELBOW290 documentation.) The degrees of freedom are *not* included when *Lab* = ALL. To constrain *all* cross-section degrees of freedom, specify *Lab* = SECT.

NEND*, *NINC

Delete constraints from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1).

Rkey

Ramping option:

OFF

Loads are step-removed (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON or FORCE

Forces on the specified degrees of freedom (*Lab*) are ramped during the next load step. The forces are ramped from the reaction forces of the previous load step, regardless of whether or not a constraint was present. If the specified node(s) and degree(s) of freedom has a force value currently defined, the force is ramped from the reaction-force value to the currently applied force value. If no force is currently applied, the force is ramped from the reaction-force value to zero. The ramping behavior is not in effect if the subsequent force is applied in tabular format.

For degrees of freedom other than structural and TEMP,[1] when performing a restart at an intermediate point during a load step,[2] the reaction-force data is not available. Therefore, the force is ramped from zero to the currently applied force value (if it exists) for the specified node(s) and degree(s) of freedom.

For structural and TEMP degrees of freedom, [1] during a restart from an intermediate point during a load step,[2] the reaction-force data is available. Therefore, it is ramped down during this restart step if no other loads are applied. See "Notes" (p. 506) for more information about the behavior of this option.

IRSTRTFLAG

Controls how the **DDELE (p. 505)** information is used during a restart analysis.

0

Retain the **DDELE (p. 505)** information for subsequent restart analysis (default).

1

Ignore the **DDELE (p. 505)** information for subsequent restart analysis.

See "Notes" (p. 506) for more information about this argument.

Notes

Deleting a constraint is not the same as setting it to zero (which fixes the degree of freedom to a zero value). Deleting a constraint has the same effect as deactivating, releasing, or setting the constraint free. The node and the degree-of-freedom label must be selected (**NSEL (p. 1266)**, **DOFSEL (p. 569)**).

For structural degrees of freedom,[1] the following limitation exists when the analysis is restarted:

- If a new force is applied (**F (p. 729)**) upon restart of the load step during which **DDELE (p. 505)**,*NODE*,*DofLabel*,,,RFORCE (or ON) was issued (and *IRSTRTFLAG* = 0), the force shows a jump to the current value at the time of restart before being ramped to its final value.

Upon restart, it is good practice to allow the reaction force to ramp down to zero in a load step, then to apply new loads in the next load step.

[1] UX, UY, YZ, ROTX, ROTY, and ROTZ

[2] Not at the beginning or end of a load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you determine that a restart analysis is unnecessary, you can set *IRSTRTFLAG* = 1 to avoid the storage overhead associated with **DDELE (p. 505)** information. This flag is especially useful in an analysis involving a large model where **DDELE (p. 505)** may be specified on many nodes and degrees of freedom, but where a restart analysis will not occur.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Node Components

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Nodes

Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Nodes

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>EdgeMVP>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Node Components

Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Nodes

Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DDOPTION, *Decomp*, *NPROCPERSOL*, *NUMSOLFORLP*

Sets domain decomposer option for a distributed-memory parallel (DMP) solution.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Decomp

Controls which domain decomposition algorithm to use.

AUTO

Automatically selects the optimal domain decomposition method (default).

MESH

Decompose the FEA mesh.

FREQ

Decompose the frequency domain for harmonic analyses.

CYCHI

Decompose the harmonic indices for cyclic symmetry modal analyses.

NPROCPERSOL

Number of processes to be used for mesh-based decomposition in conjunction with each frequency solution (*Decomp* = FREQ) or harmonic index solution (*Decomp* = CYCHI). Defaults to 1. This field is ignored when *Decomp* = MESH.

NUMSOLFORLP

Number of frequency or harmonic index solutions in a subsequent linear perturbation harmonic or linear perturbation cyclic modal analysis. This field is ignored when *Decomp* = MESH.

Command Default

The optimal algorithm for domain decomposition is automatically chosen.

Notes

This command controls options related to the domain decomposition algorithm used in a distributed-memory parallel (DMP) solution to split the analysis calculations into domains, with each domain being solved on a different process.

By default, the optimal domain decomposition algorithm (MESH, FREQ, or CYCHI) is automatically chosen. When FREQ (for a harmonic analysis) or CYCHI (for a cyclic symmetry modal analysis) is automatically chosen, the *NPROCPERSOL* argument is also automatically set to a value ≥ 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The "mesh" algorithm (*Decomp* = MESH) divides the finite element mesh into domains. In this case, domains are effectively groups of elements, with one domain being solved on each process. This algorithm seeks to create evenly sized domains (that is, domains with equal numbers of elements) as well as to minimize the size of interfaces between the newly created domains. This algorithm can be used for all analysis types.

The "frequency" algorithm (*Decomp* = FREQ) divides the specified frequency range for a harmonic analysis into domains. In this case, domains are effectively groups of frequency solutions, with one domain being solved on *NPROCPERSOL* processes. If there are more processes than frequency points, some processes will remain idle during the harmonic analysis solution. This algorithm seeks to create evenly sized domains. However, if the number of processes does not divide evenly into the number of frequency solutions, the efficiency of the parallel solution will be reduced. This algorithm can only be used for harmonic analyses using the auto (**HROPT (p. 912)**,AUTO), full (**HROPT (p. 912)**,FULL), frequency-sweep (**HROPT (p. 912)**,VT), or Krylov (**HROPT (p. 912)**,KRYLOV) method.

The "cyclic" algorithm (*Decomp* = CYCHI) divides the specified list of harmonic indices for a cyclic symmetry modal analysis into domains. In this case, domains are effectively groups of cyclic harmonic indices, with one domain being solved on *NPROCPERSOL* processes. If there are more processes than harmonic indices, some processes will remain idle during the cyclic model solution. This algorithm seeks to create evenly sized domains. However, if the number of processes does not divide evenly into the number of harmonic indices, the efficiency of the parallel solution will be reduced.

For the mesh algorithm (MESH), all available processes are used. This is not necessarily the case for the frequency and cyclic algorithms (FREQ and CYCHI).

NPROCPERSOL is only used when *Decomp* = FREQ or CYCHI. It defaults to 1, which essentially means that no mesh-based domain decomposition occurs. When *NPROCPERSOL* is defined to be greater than 1, a combination of FREQ or CYCHI decomposition and MESH decomposition is employed. As an example, consider a harmonic analysis with 50 requested frequency points (**NSUBST (p. 1287)**,50) that uses distributed processing with 100 CPU cores (`-dis -np` 100). Specifying **DDOPTION (p. 508)**,FREQ,2 would lead to 50 parallel sets of calculations, each working on a different frequency point and using 2 cores for mesh-based domain decomposition (that is, 2 groups of elements per frequency).

DDOPTION (p. 508) must be issued prior to solving the first load step. Once the first load step is completed, this command cannot be used to change the domain decomposition method. The only exception is for analyses which use the linear perturbation procedure.

In a linear perturbation analysis, **DDOPTION (p. 508)** must be entered prior to the **SOLVE (p. 1822)**,ELFORM command. In addition, the number of frequency solutions (in a subsequent harmonic analysis) or harmonic index solutions (in a subsequent cyclic modal analysis) must be input via the *NUMSOLFORLP* argument to enable proper domain decomposition to occur at the **SOLVE (p. 1822)**,ELFORM stage of the linear perturbation analysis. For more information, see Linear Perturbation Analysis in the *Structural Analysis Guide*

For more information and recommendations on how to choose the domain decomposition method, see Differences in Solution Processing in the *Parallel Processing Guide*.

For the frequency and the cyclic algorithms, solution information for the harmonic frequencies (*Decomp* = FREQ) or cyclic harmonic indices (*Decomp* = CYCHI) solved by the worker processes is only written to the output files for those processes (`Jobname`*n*`.OUT`). See Differences in General Behavior in the *Parallel Processing Guide* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

DEACT

Specifies "Element birth and death" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Elem Birth/Death

/DECRYPT,*Key1*,*Key2*

Controls decryption of command input.

APDL (p. 19): Encryption/Decryption (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key1

Key to decrypt the encrypted input created by **/ENCRYPT (p. 634)**. The only valid label is PASSWORD.

Key2

Key to decrypt the encrypted input or to set the global encryption key. The following are valid inputs:

- If *Key2* = OPENSSL or is blank, then decryption commences and the previously set global encryption key is used for decryption.
- If *Key2* has a value, then that value is set as the global encryption key.
- If *Key2* = OFF, then the global encryption password previously set by the command **/DECRYPT (p. 510)**,PASSWORD,*Key2* is reset.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

When decrypting an encrypted input, /DECRYPT,PASSWORD,OPENSSL must appear as the first line of the encrypted file. The line is inserted automatically when you issue **/ENCRYPT (p. 634)** to create the encrypted file.

To read an encrypted file, enter **/DECRYPT (p. 510)**,PASSWORD,*Key2* anywhere in the standard input file to set the global encryption key. The encryption key must be set before reading in the encrypted input.

/DECRYPT (p. 510) is also valid when entered in the Command Input Window of the Mechanical APDL user interface.

See Encrypting Command Input and Other Data in the *Ansys Parametric Design Language Guide* for more information about using **/DECRYPT (p. 510)** and performing encryption and decryption.

Menu Paths

This command cannot be accessed from a menu.

DEFINE

Specifies "Data definition settings" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)
POST26 (p. 58): Status (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (Jobname.log) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Read Options

Utility Menu>List>Status>TimeHist Postproc>Variables

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*DEL, *Val1*, *Val2*

Deletes a parameter or parameters (GUI).

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Val1

Command behavior key:

ALL --

Delete all user-defined parameters, or all user-defined parameters *and* all system parameters, as specified by *Val2*.

(blank) --

Delete the parameter(s) specified by *Val2*.

***ParmName* --**

Delete a single named parameter, specified here. (*Val2* is not used in this case.)

Val2

The parameter or parameters to delete (used only when *Val1* = ALL or (blank)):

***LOC* --**

When *Val1* is (blank), specifies the location of the parameter within the Array Parameters dialog box. The location number is based on an alphabetically ordered list of all parameters in the database.

_PRM --

When *Val1* is ALL, deletes all parameters, including those named with a leading underscore (_) (except _STATUS and _RETURN). When *Val1* is (blank), deletes only those parameters named with a leading underscore (_) (except _STATUS and _RETURN).

PRM_ --

When *Val1* is (blank), deletes only those parameters named with a trailing underscore (_).

(blank) --

When *Val1* is ALL, a (blank) value for *Val2* causes *all* user-defined parameters to be deleted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This is a command generally created by the graphical user interface (GUI). It appears in the log file (`Jobname.log`) if an array parameter is deleted from within the Array Parameters dialog.

Usage examples:

- Delete all user-defined parameters: ***DEL (p. 512)**,ALL
- Delete only those user-defined parameters named with a trailing underscore: ***DEL (p. 512)**,,PRM_
- Delete all user-defined *and* all system parameters (except for _STATUS and _RETURN): ***DEL (p. 512)**,ALL,_PRM
- Delete a parameter by specifying its location within the Array Parameters dialog: ***DEL (p. 512)**,,*LOC*
- Delete a single specified parameter by name: ***DEL (p. 512)**,*ParmName* (You cannot specify more than one named parameter at a time.)

The ***DEL (p. 512)** command does not free up memory but only deletes the specified reference. For example, memory usage increases from the continued issuance of ***DEL (p. 512)** and ***DIM (p. 530)** within ***DOWHILE (p. 573)** loops, as arrays are stored in the database (memory) and ***DEL (p. 512)** removes only the *reference* to the array. To release the memory in use, issue the **SAVE (p. 1645)** command after ***DEL (p. 512)**.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

DELETE,*SET*,*Nstart*,*Nend*

Specifies sets in the results file to be deleted before postprocessing.

AUX3 (p. 61): Results Files (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SET

Specifies that sets in the results file are to be deleted.

Nstart

The first set in a results file to be deleted.

Nend

The final set in a results file to be deleted. This field is used only if deleting more than one sequential sets.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

DELETE (p. 513) is a specification command that flags sets in the results file for deletion. It should be followed by a **COMPRESS (p. 408)** command, the corresponding action command that *deletes* the specified sets.

The **DELETE (p. 513)** command is valid only in the results file editing processor (auxiliary processor AUX3).

Menu Paths

This command cannot be accessed from a menu.

/DELETE,*Fname*,*Ext*,--,*DistKey*

Deletes a file.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current Jobname.

Ext

Filename extension (eight-character maximum).

--

Unused field.

DistKey

Key that specifies which file deletion action is performed on all processes in distributed-memory parallel mode:

0 (OFF or NO)

The program performs the file deletion only on the master process (default).

1 (ON or YES)

The program performs the file deletion locally on each process.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 or BOTH

The program performs file deletion for *Fname*.*Ext* on the master process and for *FnameN*.*Ext* on all processes.

Notes

In distributed-memory parallel (DMP) mode, only the master process will delete *Fname*.*Ext* by default. However, when *DistKey* is set to 1 (or ON, or YES) or 2 (or BOTH), the command is executed by all processes. In this case, *Fname* will automatically have the process rank appended to it. This means *FnameN*.*Ext* will be deleted by all processes, where *N* is the DMP process rank. For more information see Differences in General Behavior in the *Parallel Processing Guide*.

Menu Paths

Utility Menu>File>File Operations>Delete

DELTIM, *DTIME*, *DTMIN*, *DTMAX*, *Carry*

Specifies the time step sizes to be used for the current load step.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DTIME

Time step size for this step. If automatic time stepping is used (**AUTOTS (p. 207)**), *DTIME* is the starting time substep.

If contact elements (TARGE169, TARGE170, CONTA172, CONTA174, CONTA175, or CONTA177) are used, defaults to *TIME* or *TIME*/20 (where *TIME* is the time at the end of the load step as set on the **TIME (p. 2000)** command), depending on the physics of the model. If none of these contact elements are used, defaults to *TIME*.

DTMIN

Minimum time step (if automatic time stepping is used). The program automatically determines the default based on the physics of the model.

DTMAX

Maximum time step (if automatic time stepping is used). The program automatically determines the default based on the physics of the model.

Carry

Time step carry over key:

OFF

Use *DTIME* as time step at start of each load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON

Use final time step from previous load step as the starting time step (if automatic time stepping is used).

The program automatically determines the default based on the physics of the model.

Notes

See **NSUBST (p. 1287)** for an alternative input.

Use consistent values for *DTIME* and *TIME* (**TIME (p. 2000)**). For example, using 0.9 for *DTIME* and 1.0 for *TIME* results in one time step because 1.0 (*TIME*) is divisible by .9 (*DTIME*) at most once. If you intend to load in 10 increments over a time span of 1.0, use 0.1 for *DTIME* and 1.0 for *TIME*.

The program calculates the initial incremental time so that (*EndingTime* - *StartingTime*)/*DTIME* is an integer, which may affect the initial incremental time that you specify. For example, if the starting time is 0, the ending time is 1, and the initial incremental time is 0.4, the program rounds to the nearest integer and adjusts the time to 0.33333.

For solution efficiency, specify values for all fields of this command.

Changing the time step size upon restarting an analysis during a load step is not recommended. You should only change the time step size between load steps.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step

---

DEMORPH, *ELEM*, *DIMN*, *RMSHKY*

Move nodes in selected elements to conform to structural displacements.

PREP7 (p. 22): Morphing (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Non-structural elements to which mesh movement (morph) applies. If ALL, apply morphing to all selected elements (**ESEL (p. 661)**). If ELEM = P, graphical picking is enabled. A component may be substituted for ELEM.

DIMN

Problem dimensionality. Use "2" for a 2D problem and "3" for a 3D problem (no default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMSHKY

Remesh flag option:

0

Remesh the selected non-structural regions only if mesh morphing fails.

1

Remesh the selected non-structural regions and bypass mesh morphing.

2

Perform mesh morphing only and do not remesh.

Notes

The selected elements should include only non-structural regions adjacent to structural regions. The exterior nodes of the selected elements will usually be on the boundary of the region which will have node positions displaced. For *DIMN* = 2, elements must lie on a flat plane. The **DEMORPH (p. 516)** command requires a single domain grouping of elements be provided (multiple domains of elements are not permitted). Exterior nodes will be assumed fixed (no nodes will be morphed) unless they coincide with structural nodes having nonzero displacements.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing **DEMORPH (p. 516)**.

By default (*RMSHKY* = 0), **DEMORPH (p. 516)** will remesh the selected non-structural regions entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the **DEMORPH (p. 516)** command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC (p. 921)** command. Before a structural analysis is performed for a sequentially coupled analysis, the **DEMORPH (p. 516)** command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE (p. 923)** to delete the initial conditions.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DERIV,*IR*,*IY*,*IX*,--,*Name*,--,--,*FACTA*

Differentiates a variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IY,IX

Reference numbers of variables to be operated on. *IY* is differentiated with respect to *IX*.

--

Unused field.

Name

Thirty-two character name for identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--,--

Unused fields.

FACTA

Scaling factor (positive or negative) applied as shown below (defaults to 1.0).

Notes

Differentiates variables according to the operation:

IR = *FACTA* x d(*IY*)/d(*IX*)

Variable *IX* must be in ascending order.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Derivative

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DESIZE, *MINL*, *MINH*, *MXEL*, *ANGL*, *ANGH*, *EDGMN*, *EDGMX*, *ADJF*, *ADJM*

Controls default element sizes.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MINL

Minimum number of elements that will be attached to a line when using lower-order elements (defaults to 3 elements per line). If *MINL* = DEFA, all arguments will be set back to default values. If *MINL* = STAT, list status of command (Including on/off status). If *MINL* = OFF, deactivate default element sizing. If *MINL* = ON, reactivate default element sizing.

MINH

Minimum number of elements that will be attached to a line when using higher-order elements. Defaults to 2 elements per line.

MXEL

Maximum number of elements that will be attached to a single line (lower or higher-order elements). Defaults to 15 elements per line for h-elements. To deactivate this limit, specify a large number (such as 9999).

ANGL

Maximum spanned angle per lower-order element for curved lines. Defaults to 15 degrees per element.

ANGH

Maximum spanned angle per higher-order element for curved lines. Defaults to 28 degrees per element.

EDGMN

Minimum element edge length. Defaults to no minimum edge length. The *MINL* or *MINH* argument can override this value.

EDGMX

Maximum element edge length. Defaults to no maximum edge length. The *MXEL* argument can override this value.

ADJF

Target aspect ratio for adjacent line. Used only when free meshing. Defaults to 1.0, which attempts to create equal-sided h-elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ADJM

Target aspect ratio for adjacent line. Used only when map meshing. Defaults to 4.0, which attempts to create rectangular h-elements.

Command Default

Default settings as described for each argument are used.

Notes

DESIZE (p. 519) settings are usually used for mapped meshing. They are also used for free meshing if SmartSizing is turned off (**SMRTSIZE (p. 1811)**,OFF), which is the default. Even when SmartSizing is on, some **DESIZE (p. 519)** settings (such as maximum and minimum element edge length) can affect free mesh density. The default settings of the **DESIZE (p. 519)** command are used only when no other element size specifications (**KESIZE (p. 978)**, **LESIZE (p. 1041)**, **ESIZE (p. 668)**) exist for a certain line.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Other

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Default Element Sizing Options

---

DESOL, *ELEM*, *NODE*, *Item*, *Comp*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Defines or modifies solution results at a node of an element.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element number for which results are defined or modified. If ALL, apply to all selected elements (**ESEL (p. 661)**).

NODE

Node of element (actual node number, not the position) to which results are specified. If ALL, specify results for all selected nodes (**NSEL (p. 1266)**) of element. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Item

Label identifying results. Valid item labels are shown in Table 105: DESOL - Valid Item and Component Labels (p. 521) below. Some items also require a component label (*Comp*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Comp

Component of the item (if required); see Table 105: DESOL - Valid Item and Component Labels (p. 521).

V1

Value (in the element coordinate system) assigned to the database item (and component, if any). If zero, a zero value will be assigned. If blank, value remains unchanged.

V2, V3, V4, . . . , V6

Additional values (if any) assigned to the remaining components (in the order corresponding to the *Comp* list shown below) for the specified *Item* (starting from the specified *Comp* label and proceeding to the right).

Notes

The **DESOL (p. 520)** command defines or modifies solution results in the database at a node of an area or volume element. For example, **DESOL (p. 520)**,35,50,S,X,1000,2000,1000 assigns values 1000, 2000, and 1000 to SX, SY, and SZ (respectively) of node 50 of element 35.

The settings of the POST1 **FORCE (p. 772)**, **SHELL (p. 1784)**, and **LAYER (p. 1015)** commands, if applicable, further specify which database items are affected.

For layered composite shells, specify the current element layer (**LAYER (p. 1015)**) *before* issuing the **DESOL (p. 520)** command.

All data is stored in the solution coordinate system but is displayed in the results coordinate system (**RSYS (p. 1639)**). To list the current results, use the **PRESOL (p. 1481)** command.

Modified solution results are not saved automatically. To save separate records of modified results, use either the **RAPPND (p. 1570)** or **LCWRITE (p. 1031)** command.

Result items are available depending on element type; check the individual element for availability. Valid item and component labels for element results are:

Table 105: DESOL - Valid Item and Component Labels

Item	Comp	Description
ELEM		Element number.
S	X, Y, Z, XY, YZ, XZ	Component stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density.
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
TG	X, Y, Z	Component thermal gradient.
TF	X, Y, Z	Component thermal flux.
PG	X, Y, Z	Component pressure gradient.
EF	X, Y, Z	Component electric field.
D	X, Y, Z	Component electric flux density.
H	X, Y, Z	Component magnetic field intensity.
B	X, Y, Z	Component magnetic flux density.
CG	X, Y, Z	Concentration gradient
DF	X, Y, Z	Diffusion flux density
FMAG	X, Y, Z	Component electromagnetic force.
F	X, Y, Z	X, Y, or Z structural force.
M	X, Y, Z	X, Y, or Z structural moment.
HEAT		Heat flow.
FLOW		Fluid flow.
AMPS		Current flow.
FLUX		Magnetic flux.
CSG	X, Y, Z	X, Y, or Z magnetic current segment component.
RATE		Diffusion flow rate

Menu Paths

Main Menu>General Postproc>Define/Modify>Elem Results

DETAB, *ELEM*, *Lab*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Modifies element table results in the database.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element for which results are to be modified. If ALL, modify all selected elements (**ESEL (p. 661)**) results. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab

Label identifying results. Valid labels are as defined with the **ETABLE (p. 687)** command. Issue **ETABLE (p. 687)**,STAT to display labels and values.

V1

Value assigned to this element table result in the database. If zero, a zero value will be assigned. If blank, value remains unchanged.

V2, V3, V4, . . . , V6

Additional values (if any) assigned to consecutive element table columns.

Notes

Modifies element table (**ETABLE (p. 687)**) results in the database. For example, **DETAB (p. 522)**,35,ABC,1000,2000,1000 assigns 1000, 2000, and 1000 to the first three table columns starting with label ABC for element 35. Use the **PRETAB (p. 1490)** command to list the current results. After deleting a column of data using **ETABLE (p. 687)**,*Lab*,ERASE, the remaining columns of data are not shifted to compress the empty slot. Therefore, the user must allocate null (blank) values for *V1*, *V2*...*V6* for any ETABLE entries which have been deleted by issuing **ETABLE (p. 687)**,*Lab*,ERASE. All data are stored in the solution coordinate system but will be displayed in the results coordinate system (**RSYS (p. 1639)**).

Menu Paths

Main Menu>General Postproc>Define/Modify>ElemTabl Data

/DEVICE, *Label*, *KEY*

Controls graphics device options.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Device function label:

BBOX

Bounding box mode. For PowerGraphics plots involving elements with **/SHOW (p. 1785)**,x11 and **/SHOW (p. 1785)**,win32, Mechanical APDL generally displays dynamic rotations faster. If KEY = 1 (ON), then a bounding box (not the elements) encompassing the model is displayed and rotated, rather than the element outlines (ON is default in preprocessing). When KEY = 0 (OFF), then dynamic rotations may be slower (Mechanical APDL redraws the element outlines) for plots involving elements with **/SHOW (p. 1785)**,x11 and **/SHOW (p. 1785)**,win32. OFF is default in postprocessing. This command is ignored if **/EDGE (p. 594)**,WN,1 is set for any WN. This is ignored in POST1 and SOLUTION plots.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For any PowerGraphics plots involving elements, regardless of **/SHOW (p. 1785)** settings, plots will generally be displayed faster.

VECTOR

Vector mode. In vector mode, areas, volumes, elements, and postprocessing display geometries are shown as outlines (wireframes). When vector mode is off (default), these entities are shown filled with color.

DITHER

When dithering is turned on (default), color intensity transitions are smoothed. This selection a

applies only to smooth-shaded images, that is, Z-buffered (**/TYPE (p. 2034)**), or raster plots with Gouraud or Phong shading (**/SHADE (p. 1783)**).

ANIM

Select the animation type used on 2D devices on the PC platform. A *KEY* value of BMP (or 0) sets animation mode to the Mechanical APDL animation controller (default). A *KEY* value of AVI (or 2) sets animation mode to AVI movie player file.

FONT

Font selection for the Mechanical APDL graphics window. When *Label* = FONT, the command format is: **/DEVICE (p. 523)**,FONT,*KEY*,*Val1*,*Val2*,*Val3*,*Val4*,*Val5*,*Val6* where *KEY* determines the type of font being controlled, and values 1 through 6 control various font parameters. *These values are device specific; using the same command input file* (**/INPUT (p. 948)**) *on different machines may yield different results.*. The following *KEY* values determine the font information that will be supplied to the appropriate driver (for example, Postscript, X11, Win32, JPEG, ...):

***KEY* = 1**

The command controls the LEGEND (documentation column) font.

***KEY* = 2**

The command controls the ENTITY (node and keypoint number) font.

***KEY* = 3**

The command controls the ANNOTATION/GRAPH font.

Linux:

Values 1 through 4 are used to find a match in the X11 database of font strings. Values 1, 2, and 3 are character strings; value 4 is a nonzero integer:

Val1

Family name (for example, Courier). If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val2

Weight (for example, medium)

Val3

Slant (for example, r)

Val4

Pixel size (for example, 14). Note that this value does no affect the annotation fonts (*KEY* = 3). Use the **/TSPEC (p. 2024)** command for annotation font size.

Val5

unused

Val6

unused

PC:

The values are encoded in a PC logical font structure. Value 1 is a character string, and the remaining values are integers:

Val1

Family name (for example, Courier*New) Substitute an asterisk (*) for any blank character that appears in a family name. If *Val1* = MENU, all other values are ignored and a font selection menu appears (GUI must be active). When this value is blank, Mechanical APDL uses the first available resource it finds.

Val2

Weight (0 - 1000)

Val3

Orientation (in tenths of a degree)

Val4

Height (in logical units)

Val5

Width (in logical units)

Val6

Italics (0 = OFF, 1 = ON)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TEXT

Text size specification for the Mechanical APDL graphics window. Using this label with the **/DEVICE (p. 523)** command requires the following form: **/DEVICE (p. 523)**,TEXT,*KEY*,*PERCENT*. *KEY* = 1 for LEGEND fonts; *KEY* = 2 for ENTITY fonts. *PERCENT* specifies the new text size as a percent of the default text size. If *PERCENT* = 100, the new text size is precisely the default size. If *PERCENT* = 200, the new text size is twice the default text size.

KEY

Control key:

OFF or 0

Turns specified function off.

ON or 1

Turns specified function on or designates the LEGEND font.

2

Designates the ENTITY font.

3

Designates the ANNOTATION/GRAPH font.

Command Default

Vector mode off (that is, raster mode); dithering on.

Notes

This command is valid in any processor.

The **/DEVICE (p. 523)**,BBOX command is ignored in POST1 and SOLUTION plots. Also, the elements are displayed and rotated if you use **/DEVICE (p. 523)**,BBOX,ON and **/EDGE (p. 594)**,WN,1,ANGLE (effectively ignoring the BBOX option).

Menu Paths

Utility Menu>PlotCtrls>Device Options

Utility Menu>PlotCtrls>Font Controls>Entity Font

Utility Menu>PlotCtrls>Font Controls>Legend Font

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/DFLAB,*DOF*,*DispLab*,*ForceLab*

Changes degree-of-freedom labels for user custom elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

DOF

Number between 1 and 32 indicating which degree of freedom is to have its labels changed. For a list of these quantities, see the degree-of-freedom table in the `echprm.inc` file. The first few quantities follow:

1 = UX,FX

2 = UY,FY

3 = UZ,FZ

4 = ROTX,MX

DispLab

New label (four-character maximum) for the displacement label. The prior label is no longer valid.

ForceLab

New label (four-character maximum) for the force label for this degree of freedom. The prior label is no longer valid.

Notes

The **/DFLAB (p. 527)** command is rarely used. Use it if you are writing a custom element and want to use degrees of freedom that are not part of the standard element set.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DFLX, *NODE*, *BX*, *BY*, *BZ*, *BX2*, *BY2*, *BZ2*

Imposes a uniform magnetic flux B on an edge-element electromagnetic model.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Nodes at which the edge-flux (AZ) constraints corresponding to the uniform magnetic flux are to be specified. Valid options are ALL (default) or Component Name. If ALL, constraints are applied to all selected nodes (**NSEL (p. 1266)**).

BX, BY, BZ

Real components of magnetic flux B.

BX2, BY2, BZ2

Imaginary components of magnetic flux B.

Notes

The **DFLX (p. 528)** command sets the constraints on the edge-flux (AZ) degrees of freedom to produce a uniform magnetic flux B in an edge-based electromagnetic analysis using one of these element types: SOLID226, SOLID227, SOLID236, or SOLID237. The command ignores the corner nodes of the elements (even if they were selected) and imposes the AZ-constraints on the mid-side nodes only. The AZ-constraints are imposed in the active Cartesian coordinate system. A non-Cartesian coordinate system will be ignored by the **DFLX (p. 528)** command.

The edge-flux constraints at the mid-side nodes are derived from the magnetic vector potential **A**, which is related to the imposed magnetic flux **B** as follows:

$$A=\frac{1}{2}\left[B\times r\right]$$

where **r** is the position of the mid-side node.

The **DFLX (p. 528)** command creates a component named _DFLX for the constrained midside nodes. You can use this component to delete the constraints imposed by the **DFLX (p. 528)** command.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DFSWAVE, *KCN*, *RADIUS*, *PSDREF*, *DENS*, *SONIC*, *INCANG*, *NPARA*, *SampOpt*

Specifies the incident planar waves with random phases for a diffuse sound field.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Local coordinate system:

N

Coordinate system number. Default = 0.

DELETE

Delete defined incident diffused planar waves.

RADIUS

Radius of the reference sphere on which the incident planar waves are distributed with equal energy. Defaults to 50 x the half-maximum dimension of the structural panel.

PSDREF

Reference power spectral density. Default = 1.

DENS

Mass density of incident planar wave media. Default = 1.2041 kg/m^3.

SONIC

Sound speed in incident planar wave media. Default = 343.24 m/s)

INCANG

Maximum incident angle (0^o <= *degree* <= 180^o) against the positive z axis in the local coordinate system *KCN*. Default = 89^o.

NPARA

Number of divisions on the reference sphere with cutting planes parallel to the x-y coordinate plane of the local coordinate system. Default = 20.

SampOpt

Random sampling option:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Initializes the random generator of incident planar wave phases and samples the phases at each solving frequency.

MULT

Initializes the random generator of incident planar wave phases at the first frequency and samples the phases at each solving frequency.

MONO

Initializes the random generator of incident planar wave phases and samples the phases only once at first solving frequency so that the same phases are used over the whole frequency range for each incident planar wave.

Notes

Issue the **DFSWAVE (p. 529)** command to activate a diffuse sound field. (The **AWAVE (p. 213)** command does not activate a diffuse sound field.)

The SURF154 surface element must be defined on the surface of the structural solid element for the excitation.

The acoustic elements and the absorbing boundary condition must be defined in the open acoustic domain. Do not define the acoustic domain on the excitation side.

The **PRAS (p. 1464)** and **PLAS (p. 1387)** commands calculate the average transmission loss for multiple sampling phases at each frequency over the frequency range.

The symmetry of a panel structure cannot be used to reduce the simulation size, as the incident plane waves have varying random phase angles. The z axis of the Cartesian coordinate system (*KCN*) must be consistent with the panel's outward normal unit vector at the center of the panel's sending side.

Menu Paths

This command cannot be accessed from a menu.

***DIM**,*Par*,*Type*,*IMAX*,*JMAX*,*KMAX*,*Var1*,*Var2*,*Var3*,*CSYSID*

Defines an array parameter and its dimensions.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

Name of parameter to be dimensioned. See ***SET (p. 1720)** for name restrictions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type

Array type:

ARRAY --

Arrays are similar to standard FORTRAN arrays (indices are integers) (default). Index numbers for the rows, columns, and planes are sequential values beginning with one. Used for 1-, 2-, or 3D arrays.

ARR4 --

Same as ARRAY, but used to specify 4-D arrays.

ARR5 --

Same as ARRAY, but used to specify 5-D arrays.

CHAR --

Array entries are character strings (up to 8 characters each). Index numbers for rows, columns, and planes are sequential values beginning with one.

TABLE --

Array indices are real (non-integer) numbers which must be defined when filling the table. Index numbers for the rows and columns are stored in the zero column and row "array elements" and are initially assigned a near-zero value. Index numbers must be in ascending order and are used only for retrieving an array element. When retrieving an array element with a real index that does not match a specified index, linear interpolation is done among the nearest indices and the corresponding array element values (***SET (p. 1720)**). Used for 1-, 2-, or 3D tables.

TAB4 --

Same as TABLE, but used to specify 4-D tables.

TAB5 --

Same as TABLE, but used to specify 5-D tables.

STRING --

Array entries are character strings (up to IMAX each). Index numbers for columns and planes are sequential values beginning with 1. Row index is character position in string.

IMAX

Extent of first dimension (row). (For *Type* = STRING, *IMAX* is rounded up to the next multiple of eight and has a limit of 248). Default = 1.

JMAX

Extent of second dimension (column). Default = 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KMAX

Extent of third dimension (plane). Default = 1.

Var1

Variable name corresponding to the first dimension (row) for *Type* = TABLE, TAB4, or TAB5. Default = Row.

Var2

Variable name corresponding to the second dimension (column) for *Type* = TABLE, TAB4, or TAB5. Default = Column.

Var3

Variable name corresponding to the third dimension (plane) for *Type* = TABLE, TAB4, TAB5. Default = Plane.

CSYSID

An integer corresponding to the coordinate system ID number. Default = 0 (global Cartesian).

Notes

Up to three dimensions (row, column, and plane) may be defined using ARRAY and TABLE. Use ARR4, ARR5, TAB4, and TAB5 to define up to five dimensions (row, column, plane, book, and shelf). An index number is associated with each row, column, and plane. For array and table type parameters, element values are initialized to zero. For character and string parameters, element values are initialized to (blank). A defined parameter must be deleted (***SET (p. 1720)**) before its dimensions can be changed. Scalar (single valued) parameters should not be dimensioned. ***DIM (p. 530)**,A,,3 defines a vector array with elements A(1), A(2), and A(3). ***DIM (p. 530)**,B,,2,3 defines a 2x3 array with elements B(1,1), B(2,1), B(1,2), B(2,2), B(1,3), and B(2,3). Use ***STATUS (p. 1863)**,*Par* to display elements of array *Par*. You can write formatted data files (tabular formatting) from data held in arrays through the ***VWRITE (p. 2174)** command.

If you use table parameters to define boundary conditions, then *Var1*, *Var2*, and/or *Var3* can either specify a primary variable (listed in Table 106: *DIM - Primary Variables (p. 533)) or can be an independent parameter. If specifying an independent parameter, then you must define an additional table for the independent parameter. The additional table must have the same name as the independent parameter and may be a function of one or more primary variables or another independent parameter. All independent parameters must relate to a primary variable.

Tabular load arrays can be defined in both global Cartesian (default), cylindrical, spherical, or local (see below) coordinate systems by specifying *CSYSID*, as defined in **LOCAL (p. 1059)**. Coordinate system *CSYSID* must exist prior to issuing the ***DIM (p. 530)** command.

The following constraints apply when specifying a local coordinate system for your tabular loads:

- Only Cartesian, cylindrical and spherical coordinate systems are supported
- Angle values for Y in cylindrical or spherical coordinate systems must be input in degrees and must be positive values between 0 and 360 degrees ($0 \le Y \le 360$)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Angle values for Z in spherical coordinate system must be input in degrees and must be positive values between -90 and +90 (-90 ≤ Z ≤ 90)

If specifying a 4- or 5-D array or table, four additional fields (*LMAX*, *MMAX*, *Var4*, and *Var5*) are available. Thus, for a 4-D table, the command syntax would be:

```
*DIM,Par,Type,IMAX,JMAX,KMAX,LMAX,Var1,Var2,Var3,Var4,CSYSID
```

For a 5-D table, the command syntax is:

```
*DIM,Par,Type,IMAX,JMAX,KMAX,LMAX,MMAX,Var1,Var2,Var3,Var4,Var5,CSYSID
```

You cannot create or edit 4- or 5-D arrays or tables via the GUI.

For more information, see Array Parameters.

Table 106: *DIM - Primary Variables

Primary Variable	Label for *Var1, Var2, Var3, Var4, Var5*
Time	TIME
Frequency	FREQ
X-coordinate location	X
Y-coordinate location	Y
Z-coordinate location	Z
Temperature	TEMP
Velocity	VELOCITY
Pressure	PRESSURE [1]
Geometric gap/penetration	GAP
Cyclic sector number	SECTOR
Multistage Harmonic Index	MSHI
Harmonic-based load[a]	NHINDEX
Amplitude of the rotational velocity vector	OMEGS
Eccentricity	ECCENT
Phase shift	THETA
Element number	ELEM
Node number	NODE
Concentration	CONC

[a] Used in a Harmonic Balance Method (HBM) analysis (see Applying Loads and Constraints in an HBM Analysis in the *Harmonic Balance Method Analysis Guide*).

1. Specify PRESSURE as the independent variable (not PRES).

The X, Y, and Z coordinate locations listed above are valid in global Cartesian, or local (Cartesian, cylindrical and spherical) coordinate systems. The VELOCITY label is applicable only to the calculated fluid velocity in element FLUID116.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using PRESSURE as a primary variable, the underlying element must have the pressure DOF associated with it, or it must be a supported contact element.

The gap/penetration label (GAP) is only used for defining certain contact element real constants.

The frequency label (FREQ) is valid for harmonic analyses only.

The node and element labels (NODE and ELEM) allow you to use node and element numbers as primary variables, and their axis values should be integers.

The OMEGS, ECCENT, and THETA primary variables only apply to the COMBI214 element. The amplitude of the rotational velocity (OMEGS) is an absolute value, so only positive values of OMEGS are valid. The eccentricity (ECCENT) and phase shift (THETA) labels are only valid for nonlinear analyses.

If you use table parameters to define boundary conditions, the table names (*Par*) must not exceed 32 characters.

In thermal analyses, if you apply tabular loads as a function of temperature but the rest of the model is linear (for example, includes no temperature-dependent material properties or radiation), you should turn on Newton-Raphson iterations (**NROPT (p. 1261)**,FULL) to evaluate the temperature-dependent tabular boundary conditions correctly.

When tabular input is used to define a load via the **F (p. 729)** or **D (p. 483)** commands, the applied load may be incorrect in the following situation. This occurs when multiple solutions are performed such that the solution processor is entered multiple times (that is, the command sequence **/SOLU (p. 1821)**, **SOLVE (p. 1822)**, **FINISH (p. 758)** is repeated) and the **SET (p. 1724)** command is used in the general postprocessor (**/POST1 (p. 1461)**) to read the results file (`.rst`) between solutions. This is because the applied load becomes a fixed value in the `.rst` file and cannot be changed to the values defined by the tabular input. The workarounds are to avoid reading the `.rst` file between solutions or reissue the **F (p. 729)** or **D (p. 483)** command that uses tabular input in each solution step.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Parameters>Define/Edit

/DIRECTORY,*StrArray*,*FileName*,*Ext*,*Dir*

Put the file names in the current directory into a string parameter array.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

StrArray

Name of the "string array" parameter which will hold the returned values. String array parameters are similar to character arrays, but each array element can be as long as 128 characters. If the string

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parameter does not exist, it will be created. The array will be created as: *DIM,StrArray,STRING,64,2,numFileName

FileName

File name (64 characters maximum). Only files matching this name will be returned. The *FileName* ALL may match any file name.

Ext

File name extension (8 characters maximum). Only files with an extension matching this name will be returned. A blank or ALL will match any extension.

Directory

The directory in which the files reside. The default is the current working directory.

Notes

The **/DIRECTORY (p. 534)** command gets the file names in the current directory and puts them into a string parameter array. Each file will be included in the array as a name-extension pair.

Menu Paths

This command cannot be accessed from a menu.

DISPLAY

Specifies "Display settings" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Plot Results

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/DIST, *WN*, *DVAL*, *KFACT*

Specifies the viewing distance for magnifications and perspective.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

DVAL

Distance along the view line from the observer to the focus point (defaults to value producing full-window display). Distances "too close" to the object will produce excessive magnifications. If *DVAL* = AUTO, zero, or blank, the program will calculate the distance automatically. If *DVAL* = USER, the distance of last display will be used (useful when last display automatically calculated distance).

KFACT

DVAL interpretation key:

0

Interpret numerical *DVAL* values as described above.

1

Interpret *DVAL* as a multiplier on the current distance (*DVAL* of 2 gives twice the current distance; 0.5 gives half the current distance, etc.).

Command Default

Distance is automatically calculated to produce full window magnification.

Notes

The scale factor is relative to the window shape. For example, for objects centered in a square window and with parallel projection (no perspective), a distance of $\ell/2$ (+10%) produces a full window magnification, where ℓ is the largest in-plane vertical or horizontal dimension. See also **/AUTO (p. 207)** and **/USER (p. 2061)** commands.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Magnification

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DJ, *ELEM*, *LABEL*, *VALUE*

Specifies boundary conditions on the components of relative motion of a joint element.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element number or ALL to be specified.

LABEL

Valid labels are:

UX

Displacement in local x direction.

UY

Displacement in local y direction.

UZ

Displacement in local z direction.

ROTX

Rotation about local x axis.

ROTY

Rotation about local y axis.

ROTZ

Rotation about local y axis.

VELX

Linear velocity in local x direction.

VELY

Linear velocity in local y direction.

VELZ

Linear velocity in local z direction.

OMGX

Angular velocity in local x direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OMGY

Angular velocity in local y direction.

OMGZ

Angular velocity in local z direction.

ACCX

Linear acceleration in local x direction.

ACCY

Linear acceleration in local y direction.

ACCZ

Linear acceleration in local z direction.

DMGX

Angular acceleration in local x direction.

DMGY

Angular acceleration in local y direction.

DMGZ

Angular acceleration in local z direction.

VALUE

Value of the label.

Notes

This command is valid for MPC184 joint elements. See **DJDELE (p. 539)** for information about deleting boundary conditions applied via this command.

You can apply only one displacement, velocity, or acceleration load at any relative degree of freedom. If multiple loads are specified, the last applied load overrides the previous ones. For example, the following commands apply loads to element 100:

D,100,UX,*Value*
D,100,VELX,*Value*

In this case, the velocity load (VELX) applied in the last command will override the displacement load (UX).

Tabular boundary conditions (*VALUE* = %*tabname*%) can be used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

%_FIX% is a Mechanical APDL reserved table name. When *VALUE* is set to %_FIX%, the program sprescribe the degree of freedom to the current relative displacement value. This option is only valid for the following labels: UX, UY, UZ, ROTX, ROTY, ROTZ. In most cases, %_FIX% usage is efficient and recommended for all structural degrees of freedom.

In a modal analysis, the values of the eigenvectors at the degree of freedom connected via **DJ (p. 537)** may be insufficiently accurate to satisfy the **DJ (p. 537)** constraint conditions.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Joint Elems

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Joint Elems

---

DJDELE, *ELEM*, *LAB*

Deletes boundary conditions on the components of relative motion of a joint element.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element number or ALL. ALL (or leaving this field blank) will delete all joint element boundary conditions specified by *LAB*.

LAB

Valid labels are:

UX

Displacement in local x direction.

UY

Displacement in local y direction.

UZ

Displacement in local z direction.

ROTX

Rotation about local x axis.

ROTY

Rotation about local y axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ROTZ

Rotation about local z axis.

VELX

Linear velocity in local x direction.

VELY

Linear velocity in local y direction.

VELZ

Linear velocity in local z direction.

OMGX

Angular velocity in local x direction.

OMGY

Angular velocity in local y direction.

OMGZ

Angular velocity in local z direction.

ACCX

Linear acceleration in local x direction.

ACCY

Linear acceleration in local y direction.

ACCZ

Linear acceleration in local z direction.

DMGX

Angular acceleration in local x direction.

DMGY

Angular acceleration in local y direction.

DMGZ

Angular acceleration in local z direction.

ALL, or (blank)

Delete all applied boundary conditions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is valid for MPC184 joint elements. See **DJ (p. 537)** for information on specifying boundary conditions on the components of relative motion of a joint element.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On Joint Elems

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Joint Elems

Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On Joint Elems

Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Joint Elems

---

DJLIST, *Elem*

Lists boundary conditions applied to joint elements.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Elem

Element number or ALL (or blank). Lists joint element boundary conditions on the specified element(s).

Notes

This command is valid for MPC184 joint elements. See **DJ (p. 537)** for information on specifying boundary conditions on joint elements.

Menu Paths

Utility Menu>List>Loads>Joint Element DOF Constraints>On Picked Element

---

DK, *KPOI*, *Lab*, *VALUE*, *VALUE2*, *KEXPND*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Defines DOF constraints at keypoints.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

Keypoint at which constraint is to be specified. If ALL, apply to all selected keypoints (**KSEL (p. 995)**). If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab

Valid degree of freedom label. If ALL, use all appropriate labels *except* HDSP. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping); HDSP (hydrostatic pressure). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Acoustic labels: PRES (pressure); UX, UY, or UZ (displacements for FSI coupled elements). Electric labels: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential). Diffusion labels: CONC (concentration).

VALUE

Degree of freedom value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent signs (%) (for example, **DK (p. 541)**,NODE,TEMP,%*tabname*%). Use the ***DIM (p. 530)** command to define a table.

VALUE2

Second degree of freedom value (if any). If the analysis type and the degree of freedom allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

KEXPND

Expansion key:

0

Constraint applies only to the node at this keypoint.

1

Flags this keypoint for constraint expansion.

Lab2, Lab3, Lab4, . . . , Lab6

Additional degree of freedom labels. The same values are applied to the keypoints for these labels.

Notes

A keypoint may be flagged using *KEXPND* to allow its constraints to be expanded to nodes on the attached solid model entities having similarly flagged keypoint constraints. Constraints are transferred from keypoints to nodes with the **DTRAN (p. 583)** or **SBCTRAN (p. 1647)** commands. The expansion uses interpolation to apply constraints to the nodes on the lines between flagged keypoints. If all keypoints of an area or volume region are flagged *and the constraints (label and values) are equal*, the constraints are applied to the interior nodes of the region. See the **D (p. 483)** command for a description of nodal constraints.

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following degree of freedom labels: Electric (VOLT), structural (UX, UY, UZ, ROTX, ROTY, ROTZ), Acoustic (PRES, UX, UY, UZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).

Constraints specified by the **DK (p. 541)** command can conflict with other specified constraints. See Resolution of Conflicting Constraint Specifications in the *Basic Analysis Guide* for details.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>TimeInt>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>BasePSD>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>BasePSD>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Keypoints

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>TimeInt>On Keypoints

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>ScalarPot>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>BasePSD>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtBas>On Keypoints

Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Keypoints

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>BasePSD>On Keypoints

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtBas>On Keypoints

DKDELE, *KPOI*, *Lab*

Deletes DOF constraints at a keypoint.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

Keypoint for which constraint is to be deleted. If ALL, delete for all selected keypoints (**KSEL (p. 995)**). If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab

Valid degree of freedom label. If ALL, use all appropriate labels. Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); WARP (warping). Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). Acoustic labels: PRES (pressure); UX, UY, or UZ (displacements for FSI coupled elements). Electric label: VOLT (voltage). Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential). Diffusion label: CONC (concentration).

Notes

Deletes the degree of freedom constraints (and all corresponding finite element constraints) at a keypoint. See the **DDELE (p. 505)** command for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All KPs

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>TimeInt>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Keypoints

Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All KPs

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>TimeInt>On Keypoints

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Keypoints

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Keypoints

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>ScalarPot>On Keypoints

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Keypoints

Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Keypoints

Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DKLIST, *KPOI*

Lists the DOF constraints at keypoints.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

List constraints for this keypoint. If ALL (default), list for all selected keypoints (**KSEL (p. 995)**). If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Notes

Listing applies to the selected keypoints (**KSEL (p. 995)**) and the selected degree of freedom labels (**DOFSEL (p. 569)**).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Keypoints

Utility Menu>List>Loads>DOF Constraints>On Picked KPs

DL, *LINE*, *AREA*, *Lab*, *Value1*, *Value2*

Defines DOF constraints on lines.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

Line at which constraints are to be specified. If ALL, apply to all selected lines (**LSEL (p. 1074)**). If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*.

AREA

Area containing line. The normal to the symmetry or antisymmetry surface is assumed to lie on this area. Defaults to the lowest numbered selected area containing the line number.

Lab

Symmetry label (see 1 below):

SYMM

Generate symmetry constraints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASYM

Generate antisymmetry constraints.

Mechanical APDL degree-of-freedom labels:

UX

Displacement in X direction.

UY

Displacement in Y direction.

UZ

Displacement in Z direction.

ROTX

Rotation about X axis.

ROTY

Rotation about Y axis.

ROTZ

Rotation about Z axis.

HDSP

Hydrostatic pressure.

WARP

Warping magnitude.

TEMP, TBOT, TE2, TE3, . . ., TTOP

Temperature

VOLT

Electric scalar potential (see 2).

AZ

Magnetic vector potential in Z direction.

CONC

Concentration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Applies all appropriate DOF labels *except* HDSP.

Footnotes for *Lab*:

1. Symmetry and antisymmetry constraints are generated as described on the **DSYM (p. 580)** command.
2. Setting *Lab* = VOLT and *Value1* = 0 applies the J-normal boundary condition (current density vector (J) flows normal to the line). No input is required for the J-parallel condition because it is the natural boundary condition.

Value1

Value of DOF (real part) or table name reference on the line. Valid for all DOF labels. To specify a table, enclose the table name in % signs (for example, **DL (p. 545)**,*LINE*,*AREA*,TEMP,%*tabname*%). Use the ***DIM (p. 530)** command to define a table.

Value2

For VOLT DOFs:

Actual value of the imaginary component of the degree of freedom.

Notes

You can transfer constraints from lines to nodes with the **DTRAN (p. 583)** or **SBCTRAN (p. 1647)** commands. See the **DK (p. 541)** command for information about generating other constraints at lines.

Tabular boundary conditions (*Value1* = %*tabname*%) are available only for the following degree of freedom labels: Electric (VOLT), Structural (UX, UY, UZ, ROTX, ROTY, ROTZ), Acoustic (PRES, UX, UY, UZ), and temperature (TEMP, TBOT, TE2, TE3, . . ., TTOP).

Constraints specified with this command can conflict with other specified constraints. For more information, see Resolution of Conflicting Constraint Specifications.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>PerfEC>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Voltage>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>...with Area

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>...with Area

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>PerfEC>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>J-Normal>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Voltage>On Lines

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Lines

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>On Lines

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>...with Area

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Lines

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>On Lines

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>...with Area

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Lines

Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>On Lines

---

DLDELE, *LINE*, *Lab*

Deletes DOF constraints on a line.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

Line for which constraints are to be deleted. If ALL, delete for all selected lines (**LSEL (p. 1074)**). If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*

Lab

Constraint label:

ALL

All constraints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SYMM

Symmetry constraints.

ASYM

Antisymmetry constraints.

UX

Displacement in X direction.

UY

Displacement in Y direction.

UZ

Displacement in Z direction.

ROTX

Rotation about X axis.

ROTY

Rotation about Y axis.

ROTZ

Rotation about Z axis.

WARP

Warping magnitude.

PRES

Pressure.

TEMP, TBOT, TE2, TE3, . . ., TTOP

Temperature.

VOLT

Electric scalar potential.

AZ

Magnetic vector potential in Z direction.

CONC

Concentration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Deletes the degree of freedom constraints (and all corresponding finite element constraints) on a line previously specified with the **DL (p. 545)** command. See the **DDELE (p. 505)** command for delete details.

> **Warning:**
>
> On previously meshed lines, *all* constraints on affected nodes will also be deleted, whether or not they were specified by the **DL (p. 545)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Constraint>On All Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>PerfEC>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>Voltage>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Displacement>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Temperature>On Lines

Main Menu>Solution>Define Loads>Delete>All Load Data>All Constraint>On All Lines

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>PerfEC>On Lines

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>Voltage>On Lines

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Pressure DOF>On Lines

Main Menu>Solution>Define Loads>Delete>Magnetic>Boundary>VectorPot>On Lines

Main Menu>Solution>Define Loads>Delete>Structural>Displacement>On Lines

Main Menu>Solution>Define Loads>Delete>Thermal>Temperature>On Lines

---

DLIST,*NODE1*,*NODE2*,*NINC*

Lists DOF constraints.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2*, *NINC

List constraints for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If ALL (default), *NODE2* and *NINC* are ignored and constraints for all selected nodes (**NSEL (p. 1266)**) are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*(*NODE2* and *NINC* are ignored).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Listing applies to the selected nodes (**NSEL (p. 1266)**) and the selected degree of freedom labels (**DOFSEL (p. 569)**).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Nodes

Utility Menu>List>Loads>DOF Constraints>On Picked Nodes

DLLIST, *LINE*

Lists DOF constraints on a line.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

List constraints for this line. If ALL (default), list for all selected lines (**LSEL (p. 1074)**). If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *LINE*.

Notes

Lists the degree of freedom constraints on a line previously specified with the **DL (p. 545)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>DOF Constraints>On All Lines

Utility Menu>List>Loads>DOF Constraints>On Picked Lines

---

*DMAT, *Matrix*, *Type*, *Method*, *Val1*, *Val2*, *Val3*, *Val4*, *Val5*

Creates a dense matrix.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Matrix

Name used to identify the matrix. Must be specified.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type

Matrix type:

D --

Double precision real values (default).

Z --

Complex double precision values.

I --

Integer values.

Method

Method used to create the matrix:

ALLOC --

Allocate space for a matrix (default).

RESIZE --

Resize an existing matrix to new row and column dimensions. Values are kept from the original matrix. If the dimensions specified by *Val1* (rows) and *Val2* (columns) are greater than the original matrix size, the additional entries are assigned a value of zero.

COPY --

Copy an existing matrix.

LINK --

Link to an existing matrix. The memory will be shared between the original matrix and the new matrix. This is useful for manipulating a submatrix of a larger matrix. The *Val1* through *Val5* arguments will be used to specify the lower and upper bounds of row and column numbers from the original matrix.

IMPORT --

Import the matrix from a file.

Val1, *Val2*, *Val3*, *Val4*, *Val5*

Additional input. The meaning of *Val1* through *Val5* will vary depending on the specified *Method*. See details below.

The following *Valx* fields are used with *Method* = ALLOC or *Method* = RESIZE:

Val1

Number of rows in the matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val2

Number of columns in the matrix.

Val3

Memory allocation type (used only with *Method* = ALLOC):

INCORE --

In-core memory allocation (default).

OUTOFCORE --

Out-of-core memory allocation.

The following *Valx* fields are used with *Method* = COPY:

Val1

Name of the matrix to copy.

Val2

Method used for copying the matrix:

TRANS --

Transpose the original matrix. *Val3* and *Val4* are ignored.

REAL --

Copy the real part to the output matrix. This option only applies when copying a complex value matrix to a real value matrix. *Val3* and *Val4* are ignored.

IMAG --

Copy the imaginary part to the output matrix. This option only applies when copying a complex value matrix to a real value matrix. *Val3* and *Val4* are ignored.

EXTRACT --

Extract a submatrix based on row and column numbers specified by *Val3* and *Val4*.

Val3

Name of integer vector (***VEC (p. 2085)**) containing row numbers. If no vector is specified, defaults to all rows.

Val4

Name of integer vector (***VEC (p. 2085)**) containing column numbers. If no vector is specified, defaults to all columns.

The following *Valx* fields are used with *Method* = LINK:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val1

Name of the original matrix.

Val2

First column number (defaults to 1).

Val3

Last column number (defaults to the maximum column number of the original matrix).

Val4

First row number (defaults to 1).

Val5

Last row number (defaults to the maximum row number of the original matrix).

The following table describes the *Valx* fields used with *Method* = IMPORT:

***Method* = IMPORT**					
Val1	***Val2***	***Val3***	***Val4***	***Val5***	**Description**
FULL	File name [a]	Matrix type: STIFF - Stiffness matrix MASS - Mass matrix DAMP - Damping matrix RHS - Load vector K_RE - Real part of the stiffness matrix K_IM - Imaginary part of the stiffness matrix	(not used)	(not used)	Import a matrix or load vector from an existing FULL file. For a complex stiffness matrix, you can use the K_RE or K_IM label to import only the real part or only the imaginary part. The K_RE and K_IM options both create real matrices.
MODE	File name [a]	First mode number	Last mode number	(not used)	Import a set of modes from an existing MODE file.
CMS	File name [a]	Mode type: NOR - Normal modes IRF - Inertial relief modes CST - Constraint modes	(not used)	(not used)	Import a modal basis from an existing CMS file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method = IMPORT					
Val1	***Val2***	***Val3***	***Val4***	***Val5***	**Description**
		RSD - Residual modes			
TCMS	File name [a]	Mode type: NOR - Normal modes IRF - Inertial relief modes CST - Constraint modes	(not used)	(not used)	Import a modal basis from an existing TCMS file.
RST	File name [a]	First data set number	Last data set number	Result Type: NSL - Nodal Solution VSL - Transient velocity solution ASL - Transient acceleration solution RF - Reaction Forces	Import degree of freedom results from an existing RST file.
SUB	File name [a]	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix RHS - Load vector	(not used)	(not used)	Import a matrix or load vector from an existing SUB file.
HBMAT	File name [a]	File format: ASCII	(not used)	(not used)	Import a matrix from an existing Harwell-Boeing format file.

Method = IMPORT					
Val1	***Val2***	***Val3***	***Val4***	***Val5***	**Description**
		BINARY			
EMAT	File name [a]	Matrix type: STIFF - Stiffness matrix MASS - Mass matrix DAMP - Damping matrix STRESS - Stress-stiffness matrix RHS - Load vector	Element number	(not used)	Import an element matrix or load vector from an existing EMAT file.
MAT	File name [a]	(not used)	(not used)	(not used)	Restore from a previous ***EXPORT (p. 716)** (*FORMAT* = MAT) command.
APDL	Array parameter name	(not used)	(not used)	(not used)	Import an existing array parameter.
MMF	File name [a]	(not used)	(not used)	(not used)	Import a dense matrix from an existing Matrix Market file.
DMIG	File name [a]	File Format: FREE - Free field format LARGE - Large field format (default) (see "Notes" (p. 556) for examples)	Delimiter character (only for FREE field format): ',' - comma character (default) 'S' - space character	Matrix name (default is the first matrix of the DMIG file)	Import a matrix from an existing Nastran DMIG file.

[a] File name is case-sensitive with a 32-character maximum.

Notes

This command allows you to create a dense matrix. To create a sparse matrix, use the ***SMAT (p. 1801)** command. ***SMAT (p. 1801)** is recommended for large matrices obtained from the .FULL or .HBMAT file. Refer to the **HBMAT (p. 888)** command documentation for more information about .FULL file contents.

Use the ***VEC (p. 2085)** command to create a vector.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For very large matrices, use the OUTOFCORE option (*Method* = ALLOC or COPY) to keep some of the matrix on disk if there is insufficient memory.

When importing a dense matrix from a **DMIG** file, you can define the formatting of the file using the *Val3* and *Val4* fields. Here are a few different example of formats:

- A LARGE field format file (using *Val3* = 'LARGE'):

```
...
DMIG*   KAAX                           21               2
*                     21               1-2.261491337E+08
...
```

- A FREE field format file with blank separators (using *Val4* = 'S'):

```
...
DMIG stiff 1 2 1 2 29988.
1 6 149940. 2 2 -29988.
2 6 149940.
...
```

- A FREE field format file with a comma separator (using *Val4* = ','):

```
...
DMIG,KF,22321,3,,22321,2,-5.00E+6
DMIG,KF,22320,3,,22320,2,-5.00E+6
...
```

Requirement when importing matrices from a Nastran DMIG file: To ensure that the .sub file is properly generated from matrices imported from Nastran **DMIG** file, the generalized coordinates for a CMS superelement (SPOINTS in Nastran) must appear last (have highest ID number).

Example Usage

APDL Math Examples in the *Ansys Parametric Design Language Guide*

Menu Paths

This command cannot be accessed from a menu.

DMPEXT, *SMODE*, *TMODE*, *Dmpname*, *Freqb*, *Freqe*, *NSTEPS*

Extracts modal damping coefficients in a specified frequency range.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SMODE

Source mode number. There is no default for this field; you must enter an integer greater than zero.

TMODE

Target mode. Defaults to *SMODE*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dmpname

Array parameter name containing the damping results. Defaults to d_damp.

Freqb

Beginning frequency range (real number greater than zero) or 'EIG' at eigenfrequency of source mode. 'EIG' is valid only if *SMODE* = *TMODE*. Note that EIG must be enclosed in single quotes when this command is used on the command line or in an input file. There is no default for this field; you must enter a value.

Freqe

End of frequency range. Must be blank for *Freqb* = EIG. Default is *Freqb*.

NSTEPS

Number of substeps. Defaults to 1.

Notes

DMPEXT invokes a Mechanical APDL macro that uses modal projection techniques to compute the damping force by the modal velocity of the source mode onto the target mode. From the damping force, damping parameters are extracted. DMPEXT creates an array parameter *Dmpname*, with the following entries in each row:

- response frequency
- modal damping coefficient
- modal squeeze stiffness coefficient
- damping ratio
- squeeze-to-structural stiffness ratio

The macro requires the modal displacements from the file Jobname.EFL obtained from the **RMFLVEC (p. 1612)** command. In addition, a node component FLUN must exist from all FLUID136 nodes. The computed damping ratio may be used to specify constant or modal damping by means of the **DMPRAT (p. 562)** or **MDAMP (p. 1117)** commands. For Rayleigh damping, use the **ABEXTRACT (p. 88)** command to compute ALPHAD and BETAD damping parameters. See Thin Film Analysis for more information on thin film analyses.

The macro uses the **LSSOLVE (p. 1085)** command to perform two load steps for each frequency. The first load case contains the solution of the source mode excitation and can be used for further postprocessing. Solid model boundary conditions are deleted from the model. In addition, prescribed nodal boundary conditions are applied to the model. You should carefully check the boundary conditions of your model prior to executing a subsequent analysis.

This command is also valid in PREP7.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Solution>ThinFilm>DampExtract>Eigenfrequency

Main Menu>Solution>ThinFilm>DampExtract>Frequency Range

DMPOPTION, *FileType*, *Combine*, *ResCombFreq*, *DeleOpt*

Specifies distributed-memory parallel (DMP) file combination options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FileType

Type of solution file to combine after a distributed memory parallel solution. There is no default; if (blank), the command is ignored.

RST

Results files (`.rst`, `.rth`, `.rmg`, `.rstp`)

EMAT

Element matrix files (`.emat`).

ESAV

Element saved data files (`.esav`)

MODE

Modal results files (`.mode`)

MLV

Modal load vector file (`.mlv`)

IST

Initial state file (`.ist`)

FULL

Full matrix file (`.full`)

Reduced complex displacement file (`.`)

RDSP

Reduced displacement file (`.rdsp`)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RNNN

Multiframe restart files (.r*nnn*)

Combine

Option to combine solution files.

Yes

Combine solution files (default for all files except the .r*nnn* files).

No

Do not combine solution files (default for the .r*nnn* files only).

ResCombFreq

Frequency used to combine the local results files during a distributed memory parallel solution. This option applies only when *FileType* = RST and *Combine* = YES.

NONE

Do not combine the local results files during solution. The local results files is combined only upon leaving the solution processor (default).

ALL

Combines the local results files at every time point.

LAST

Combines the local results files at the last time point of every load step.

DeleOpt

Option to delete local solution files of the type specified by *FileType* option after they are combined. This option applies only when *Combine* = Yes.

Yes

Delete the local solution files after they are combined.

No

Do not delete the local solution files after they are combined (default).

Command Default

Local solution files are automatically combined into a single global file upon leaving the solution processor (for example, Jobname*n*.rst files are combined into one Jobname.rst file). This is true for all files except the .R*nnn* files. Because they may be required in a subsequent analysis, local files are not automatically deleted after combination.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **DMPOPTION (p. 559)** command controls how solution files are written during a distributed-memory parallel (DMP) solution. This command is most useful for controlling how results files (`.rst`, `.rth`, etc.) are written.

In a distributed memory parallel solution, a local results file is written by each process (`Jobname`*n*`.ext`, where *N* is the process number). By default, the program automatically combines the local results files (for example, `Jobname`*n*`.rst`) upon leaving the solution processor (for example, upon the **FINISH (p. 758)** command) into a single global results file (`Jobname.rst`) which can be used in postprocessing.

Alternatively, the *ResCombFreq* argument can be used to combine the local results files at certain time points during the distributed solution to create a single combined results file that can be used to postprocess the model while the solution progresses. Doing so increases data communication and I/O between processes, leading to slower performance.

The *ResCombFreq* option only applies when solving static analyses, and harmonic or transient analyses that use the full method. It does not apply to mode superposition harmonic and mode superposition transient analyses. It does not apply when using the frequency domain decomposition option (**DDOPTION (p. 508)**,FREQ) in a harmonic analysis.

To reduce communication and I/O, issue **DMPOPTION (p. 559)**,RST,NO to bypass this step of combining the local results files; the local files remain on the local disks in the current working directory. You can then use **RESCOMBINE (p. 1589)** in the POST1 general postprocessor (**/POST1 (p. 1461)**) to read all results into the database for postprocessing.

The **RESCOMBINE (p. 1589)** command macro is intended for use with POST1. If you want to postprocess distributed parallel solution results using the POST26 time-history postprocessor (**/POST26 (p. 1462)**), it is recommended that you combine your local results files into one global results file (**DMPOPTION (p. 559)**,RST,YES or **COMBINE (p. 406)**).

Local `.emat`, `.esav`, `.mode`, `.mlv`, `.ist`, `.rfrq`, `.rdsp`, and `.full` files are also written (when applicable) by each process in a distributed memory parallel solution. If these files are not needed for a downstream solution or operation, you can issue the command **DMPOPTION (p. 559)**,*FileType*,NO for each file type to bypass the file combination step and thereby improve performance.

If **DMPOPTION (p. 559)**,MODE,NO or **DMPOPTION (p. 559)**,RST,NO is specified in a modal analysis, element results cannot be written to the combined mode file (`Jobname.mode`). In this case, if distributed-memory parallel processing is used in a downstream harmonic or transient analysis that uses the mode-superposition method, the *MSUPkey* on the **MXPAND (p. 1203)** command can retain its value. However, if shared-memory parallel processing is used in the downstream harmonic or transient analysis, the *MSUPkey* is effectively set to NO.

If **DMPOPTION (p. 559)**,RNNN,YES is specified, all multiframe restart files named `Jobname.r001` to `Jobname.r999` are automatically combined upon leaving the solution processor (which can be slow and inefficient). To manually combine a single set of `.r`*nnn* restart files, use the **COMBINE (p. 406)** command.

Since local solution files may be required in a downstream analysis, the option to delete them after combination is disabled (*DeleOpt* = No) by default. However, you can enable this option to reduce disk space usage by removing certain files if you know that they are not needed in any subsequent analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DMPOPTION (p. 559) can be changed between load steps; however, doing so does not affect which set of solution files are combined. The values of *Combine* and *DeleOpt* are overwritten if **DMPOPTION (p. 559)** is issued multiple times for the same *FileType*. As a result, only the last values of *Combine* and *DeleOpt* for each *FileType* upon leaving the solution processor determine whether the local solution files are combined with or without deletion. For example, in a two-load-step solution, if **DMPOPTION (p. 559)**, RST, NO is issued in the first load step and **DMPOPTION (p. 559)**, RST, YES, ,YES is issued in the second load step, all sets on the local results files will be combined, and then the local results files will be deleted. If the opposite is true (**DMPOPTION (p. 559)**, RST, YES, ,YES is issued in the first load step, and **DMPOPTION (p. 559)**, RST, NO is issued in the second load step), no results files are combined, and thus no local results files are deleted.

After using **DMPOPTION (p. 559)** to suppress file combination, you may find it necessary to combine the local files for a specific *FileType* for use in a subsequent analysis. In this case, use the **COMBINE (p. 406)** command to combine local solution files into a single, global file.

Menu Paths

This command cannot be accessed from a menu.

DMPRAT, *RATIO*

Sets a modal damping ratio.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RATIO

Modal damping ratio (for example, 2% is input as 0.02).

Command Default

Use damping as defined by Damping in the *Structural Analysis Guide*.

Notes

Sets a damping ratio for use in a mode-superposition transient analysis (**ANTYPE (p. 162)**,TRANS with **TRNOPT (p. 2017)**,MSUP), a mode-superposition harmonic analysis (**ANTYPE (p. 162)**,HARMIC with **HROPT (p. 912)**,MSUP) analysis, or a spectrum (**ANTYPE (p. 162)**,SPECTR) analysis.

DMPRAT (p. 562) can also be defined in a substructure analysis that uses component mode synthesis. The damping ratio is added on the diagonal of the reduced damping matrix, as explained in Component Mode Synthesis (CMS).

This command is also valid in PREP7.

Additional Information

Damping Matrices in the *Mechanical APDL Theory Reference*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping

Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

DMPSTR, *COEFF*, *DMPSFreqTab*

Sets constant structural damping data.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

COEFF

Constant structural damping coefficient.

DMPSFreqTab

Average excitation frequency table (Hz) for the calculation of equivalent viscous damping from structural damping input (**DMPSTR (p. 563)** and **MP (p. 1160)**,DMPS) in a full transient analysis. Enclose the table name in percent signs (%) and use the ***DIM (p. 530)** command to define the table with primary variable TIME. To define a constant frequency instead of a table, see **TRNOPT (p. 2017)**.

Command Default

Use damping as defined by Damping in the *Structural Analysis Guide*.

Notes

Sets a constant structural (or hysteretic) damping coefficient for use in these analysis types:

- Harmonic (**ANTYPE (p. 162)**,HARMIC) -- full, Krylov, or mode-superposition [1]
- Modal (**ANTYPE (p. 162)**,MODAL) with **MODOPT (p. 1140)**,UNSYM, DAMP, or QRDAMP.

1. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

DMPSTR (p. 563) is also supported in transient (**ANTYPE (p. 162)**,TRANS) analyses (full or QRDAMP mode-superposition) as an equivalent viscous damping when an average excitation frequency is specified. For a full or mode-superposition transient, specify a constant excitation frequency as *DMPSFreq* on the **TRNOPT (p. 2017)** command. For a full transient, you can alternatively specify a table of frequencies using *DMPSFreqTab* on this command. *DMPSFreqTab* overwrites *DMPSFreq* on **TRNOPT (p. 2017)**.

Note that for structures with multiple materials, **MP (p. 1160)**,DMPS can also be used to specify constant structural material damping on a per material basis. If both **DMPSTR (p. 563)** and **MP (p. 1160)**,DMPS are specified, the damping effects are additive.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping

Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

DNSOL, *NODE*, *Item*, *Comp*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*, *DataKey*

Defines or modifies solution results at a node.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node for which results are specified. If ALL, apply to all selected nodes (**NSEL (p. 1266)**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Item

Label identifying results, see Table 107: DNSOL - Valid Item and Component Labels (p. 565). Some items also require a component label.

Comp

Component of the item. Valid component labels are shown Table 107: DNSOL - Valid Item and Component Labels (p. 565) below.

V1, V2, V3, . . . , V6

Value assigned to result. If zero, a zero value will be assigned. If blank, the value remains unchanged. Additional values (if any) assigned to the remaining components (in the order corresponding to the *Comp* list shown below for the specified *Item* (starting from the specified *Comp* label and proceeding to the right).

DataKey

Key to specify which data is modified:

AUTO

Nodal-averaged results are used if available. Otherwise, the element-based data is used if available. (Default)

ESOL

Only element-based results are used. If they are not available, the command is ignored.

NAR

Only nodal-averaged results are used. If they are not available, the command is ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

DNSOL (p. 564) can be used only with FULL graphics activated (**/GRAPHICS (p. 866)**,FULL); it will not work correctly with PowerGraphics activated.

DNSOL (p. 564) defines or modifies solution results in the database at a node. For example, **DNSOL (p. 564)**,35,U,X,.001,.002,.001 assigns values 0.001, 0.002, and 0.001 to UX, UY, and UZ (respectively) for node 35. All results that are changed in the database, including the nodal degree of freedom results, are available for all subsequent operations. All data is stored in the solution coordinate system but is displayed in the results coordinate system (**RSYS (p. 1639)**). Use **PRNSOL (p. 1505)** to list the current results.

Data input by **DNSOL (p. 564)** is stored in temporary space and does not replace information in the database. Therefore, data input by this command may be overwritten if a change is made to the selected set of nodes or if an output operation acts on a new *Item*.

Issuing **DNSOL (p. 564)** requires you to place the data type (stress/strain) in the element nodal records. To work around this requirement, use the **DESOL (p. 520)** command or equivalent path to add a dummy element stress/strain record.

Result items are available depending on element type; check the individual element for availability. Valid item and component labels for element results are shown in Table 107: DNSOL - Valid Item and Component Labels (p. 565).

Using DNSOL (p. 564) with Nodal-Averaged Results

If nodal-averaged results (**OUTRES (p. 1336)**,NAR or another nodal-averaged label) are in the database, then **DNSOL (p. 564)** acts on the nodal-averaged data for the applicable items (S, EPEL, EPPL, EPCR, EPTH, EPSW) by default. You can change this behavior via the *DataKey* argument.

DNSOL (p. 564) behavior differs when the command acts on nodal-averaged results. The nodal-averaged results that are defined or modified will be apparent in subsequent command operations (for example **PRNSOL (p. 1505)**, **PLNSOL (p. 1425)**) in both full model graphics mode (**/GRAPHICS (p. 866)**,FULL) and PowerGraphics mode (**/GRAPHICS (p. 866)**,POWER). The resultant data is stored in the global Cartesian coordinate system but is displayed in the results coordinate system (**RSYS (p. 1639)**). **DNSOL (p. 564)** can only be applied to nodal-averaged results if they are already in the database; otherwise, the modifications are applied to the element-based solution in temporary memory. The modified nodal-averaged results are not saved to the results file automatically. To save separate records of modified nodal-averaged results, use **LCWRITE (p. 1031)**, **RAPPND (p. 1570)**, or **RESWRITE (p. 1604)**.

DNSOL (p. 564) can only modify component values (*Comp* = X, Y, Z, XY, YZ, or XZ) for nodal-averaged results. If you attempt to modify principal values using **DNSOL (p. 564)** with *DataKey* = AUTO, then the modification is applied to element-based results if they are available.

Table 107: DNSOL - Valid Item and Component Labels

Valid Item and Component Labels for Nodal DOF Results		
Item	**Comp**	**Description**
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP[1]		Temperature.
PRES		Pressure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Item and Component Labels for Nodal DOF Results

Item	Comp	Description
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
CONC		Concentration.

Valid Item and Component Labels for Area and Volume Element Results

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress. [2]
"	1, 2, 3	Principal stress. [3]
"	INT	Stress intensity. [3]
"	EQV	Equivalent stress. [3]
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain. [2]
"	1, 2, 3	Principal elastic strain. [3]
"	INT	Elastic strain intensity. [3]
"	EQV	Elastic equivalent strain. [3]
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain. [2]
"	1, 2, 3	Principal thermal strain. [3]
"	INT	Thermal strain intensity. [3]
"	EQV	Thermal equivalent strain. [3]
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain. [2]
"	1, 2, 3	Principal plastic strain. [3]
"	INT	Plastic strain intensity. [3]
"	EQV	Plastic equivalent strain. [3]
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain. [2]
"	1, 2, 3	Principal creep strain. [3]
"	INT	Creep strain intensity. [3]
"	EQV	Creep equivalent strain. [3]
EPSW		Swelling strain. [2]
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
SEND	ELASTIC	Elastic strain energy density. (For viscoelastic materials, the stored energy.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Item and Component Labels for Area and Volume Element Results

Item	Comp	Description
"	PLASTIC	Plastic strain energy density.
"	CREEP	Creep strain energy density.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, ..., TTOP instead of TEMP.

2. For these component values, nodal-averaged results are modified if they are available in the results file and *DataKey* = AUTO or NAR.

3. Modifying principal values of nodal-averaged results is not supported.

Menu Paths

Main Menu>General Postproc>Define/Modify>Nodal Results

***DO**,*Par*,*IVAL*,*FVAL*,*INC*

Defines the beginning of a do-loop.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

The name of the scalar parameter to be used as the loop index. See ***SET (p. 1720)** for name restrictions. Any existing parameter of the same name will be redefined. There is no character parameter substitution for the *Par* field.

IVAL*, *FVAL*, *INC

Initially assign *IVAL* to *Par*. Increment *IVAL* by *INC* for each successive loop. If *IVAL* exceeds *FVAL* and *INC* is positive, the loop is not executed. *INC* defaults to 1. Negative increments and non-integer numbers are allowed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The block of commands following the ***DO (p. 567)** command (up to the ***ENDDO (p. 636)** command) is executed repeatedly until some loop control is satisfied. Printout is automatically suppressed on all loops after the first (include a **/GOPR (p. 865)** command to restore the printout). The command line loop control (*Par*,*IVAL*,*FVAL*,*INC*) must be input; however, a Use the ***IF (p. 927)** within the block can also be used to control looping (***EXIT (p. 707)**, ***CYCLE (p. 462)**). One level of internal file switching is used for each nested ***DO (p. 567)**. Twenty levels of nested do-loops are allowed.

Note:

Do-loops that include **/INPUT (p. 948)**, ***USE (p. 2059)**, or an "Unknown Command" macro, have less nesting available because each of these operations also uses a level of file switching. The ***DO (p. 567)**, ***ENDDO (p. 636)**, and any ***CYCLE (p. 462)** and ***EXIT (p. 707)** commands for a do-loop must all be read from the same file (or keyboard). You cannot use the MULTIPRO or ***CREATE (p. 434)** commands within a ***DO (p. 567)**-loop. Picking operations should also not be used within a ***DO (p. 567)**-loop.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

DOF, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*

Adds degrees of freedom to the current DOF set.

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 569)

Lab1*, *Lab2*, *Lab3*, . . . , *Lab10

Valid labels are: UX, UY, UZ (structural displacements); ROTX, ROTY, ROTZ (structural rotations); TEMP, TBOT, TE2, TE3, . . ., TTOP (temperatures); PRES (pressure); VOLT (voltage); MAG (magnetic scalar potential); AZ (magnetic vector potential); CURR (current); EMF (electromotive force drop); CONC (concentration); DELETE.

Command Default

Use degree of freedom set determined from element types.

Notes

The degree of freedom (DOF) set for the model is determined from all element types defined. This command may be used to add to the current set. The ALL label may be used on some commands to represent all labels of the current degree of freedom set for the model. Issue the **DOF (p. 568)** command

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with no arguments to list the current set. Use the DELETE label to delete any previously added DOFs and return to the default DOF set.

Product Restrictions

Command Option Lab	Available Products
UX, UY, UZ, ROTX, ROTY, ROTZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MAG, VOLT	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
AZ	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PRES	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
TEMP, TBOT, TE2, TE3	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CURR, EMF	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CONC	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DELETE	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Menu Paths

Main Menu>Preprocessor>Element Type>Add DOF

Main Menu>Preprocessor>Element Type>Remove DOFs

DOFSEL, *Type*, *Dof1*, *Dof2*, *Dof3*, *Dof4*, *Dof5*, *Dof6*

Selects a DOF label set for reference by other commands.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set of labels.

A

Add labels to the current set.

U

Unselect (remove) labels from the current set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Restore the full set of labels.

STAT

Display the current select status.

Dof1*,*Dof2*,*Dof3*,...,*Dof6

Used only with *Type* = S, A, or U. Valid lables are:

Structural labels: UX, UY, or UZ (displacements); U (UX, UY, and UZ) ; ROTX, ROTY, or ROTZ (rotations); ROT (ROTX, ROTY, and ROTZ); DISP (U and ROT); HDSP (Hydrostatic pressure).

Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature).

Acoustic labels: PRES (pressure); UX, UY, or UZ (displacements for FSI coupled elements).

Electric labels: VOLT (voltage); EMF (electromotive force drop); CURR (current).

Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential); A (AZ); CURR (current).

Structural force labels: FX, FY, or FZ (forces); F (FX, FY, and FZ); MX, MY, or MZ (moments); M (MX, MY, and MZ); FORC (F and M); DVOL (fluid mass flow rate).

Thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow).

Fluid flow force label: FLOW (fluid flow).

Electric force labels: AMPS (current flow); CHRG (electric charge).

Magnetic force labels: FLUX (scalar magnetic flux); CSGZ (magnetic current segment).

Diffusion labels: CONC (concentration); RATE (diffusion flow rate).

Command Default

Degree of freedom (and the corresponding force) labels are determined from the model.

Notes

Selects a degree of freedom label set for reference by other commands. The label set is used on certain commands where ALL is either input in the degree of freedom label field or implied. The active label set has no effect on the solution degrees of freedom. Specified labels which are not active in the model (from the **ET (p. 686)** or **DOF (p. 568)** command) are ignored. As a convenience, a set of force labels corresponding to the degree of freedom labels is also selected. For example, selecting UX also causes FX to be selected (and vice versa). The force label set is used on certain commands where ALL is input in the force label field.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Constraints

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Forces

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Constraints

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Forces

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Constraints

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Forces

Main Menu>Solution>Define Loads>Settings>Replace vs Add>Constraints

Main Menu>Solution>Define Loads>Settings>Replace vs Add>Forces

---

DOMEGA, *DOMGX*, *DOMGY*, *DOMGZ*

Specifies the rotational acceleration of the structure.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DOMGX*, *DOMGY*, *DOMGZ

Rotational acceleration of the structure about the global Cartesian X, Y, and Z axes.

Notes

Specifies the rotational acceleration of the structure about each of the global Cartesian axes. Rotational accelerations may be defined in these analysis types:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC) -- full, VT [1], Krylov [1], or mode-superposition [2]
- Transient (**ANTYPE (p. 162)**,TRANS) -- full or mode-superposition [2]
- Substructuring (**ANTYPE (p. 162)**,SUBSTR)
- Modal (**ANTYPE (p. 162)**,MODAL)

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

See Acceleration Effect in the *Mechanical APDL Theory Reference* for details. Units are radians/time^2.

The **DOMEGA (p. 571)** command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *DOMEGA_X*, *DOMEGA_Y*, and *DOMEGA_Z* input values (***DIM (p. 530)**) for full transient and harmonic analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Related commands are **ACEL (p. 92)**, **CGLOC (p. 319)**, **CGOMGA (p. 320)**, **DCGOMG (p. 501)**, and **OMEGA (p. 1324)**.

See Analysis Tools in the *Theory Reference* for more information.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Accel>Global
Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Accel>Global
Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Accel>Global
Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Accel>Global

---

***DOT**,*Vector1*,*Vector2*,*Par_Real*,*Par_Imag*,*Conj*

Computes the dot (or inner) product of two vectors.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Vector1

Name of first vector; must have been previously specified by a ***VEC (p. 2085)** command.

Vector2

Name of second vector; must have been previously specified by a ***VEC (p. 2085)** command.

Par_Real

Parameter name that contains the result.

Par_Imag

Parameter name that contains the imaginary part of the result (used only for complex vectors).

Conj

Key to specify use of the conjugate of *Vector1* when the vectors are complex:

TRUE

Use the conjugate of *Vector1* (default).

FALSE

Do not use the conjugate of *Vector1*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

If *Vector1* and *Vector2* are complex, the complex conjugate of *Vector1* is used to compute the result (*Par_Real*, *Par_Imag*). Therefore, ***DOT** applied to complex vectors performs the operation:

$$res = V1^{*} \cdot V2$$

Set *Conj* = FALSE if you do not want to use the conjugate of *Vector1*. In this case, the operation is:

$$res = V1 \cdot V2$$

Menu Paths

This command cannot be accessed from a menu.

***DOWHILE**, *Par*

Loops repeatedly through the next *ENDDO (p. 636) command.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

The name of the scalar parameter to be used as the loop index. There is no character parameter substitution for the *Par* field.

Notes

DOWHILE (p. 573)** loops repeatedly through the next ***ENDDO (p. 636)** command as long as *Par* is greater than zero. The block of commands following the ***DOWHILE (p. 573)** command (up to the ***ENDDO (p. 636)** command) is executed repeatedly until some loop control is satisfied. Printout is automatically suppressed on all loops after the first (include a **/GOPR (p. 865)** command to restore the printout). The command line loop control (*Par*) must be input; however, ***IF (p. 927)** within the block can also be used to control looping (EXIT (p. 707)**, ***CYCLE (p. 462)**). One level of internal file switching is used for each nested ***DOWHILE (p. 573)**. Twenty levels of nested do-loops are allowed.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DSCALE, *RFACT*, *IFACT*, *TBASE*

Scales DOF constraint values.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RFACT

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

TBASE

Base temperature for temperature difference. For temperatures, the scale factor is applied to the temperature difference (*T*-*TBASE*) and then added to *TBASE*. *T* is the current temperature.

Notes

Scales degree of freedom constraint values (displacement, temperature, etc.) in the database. If velocity and acceleration boundary conditions are applied in a structural analysis, they are also scaled by this command. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note:

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

Scaling applies to the previously defined values for the selected nodes (**NSEL (p. 1266)**) and the selected degree of freedom labels (**DOFSEL (p. 569)**). Issue **DLIST (p. 550)** command to review results.

DSCALE (p. 574) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Constraints

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Constraints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/DSCALE, *WN*, *DMULT*

Sets the displacement multiplier for displacement displays.

GRAPHICS (p. 16): Scaling (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

DMULT

AUTO or 0

Scale displacements automatically so that maximum displacement (vector amplitude) displays as 5 percent of the maximum model length, as measured in the global Cartesian X, Y, or Z directions.

1

Do not scale displacements (that is, scale displacements by 1.0, true to geometry). Often used with large deflection results.

FACTOR

Scale displacements by numerical value input for FACTOR.

OFF

Remove displacement scaling (that is, scale displacements by 0.0, no distortion).

USER

Set *DMULT* to that used for last display (useful when last *DMULT* value was automatically calculated).

Command Default

The default value is AUTO or 0 except when:

Large deflection effects are included (**NLGEOM (p. 1231)**,ON) and it is not a modal analysis; then the default is 1.

It is a spectrum analysis (**ANTYPE (p. 162)**,SPECTR); then the default is OFF.

The amplitude of a time-harmonic solution is computed using the **HRCPLX (p. 910)** command (OMEGAT $\geq$ 360°); then the default is OFF.

The amplitude of a complex modal or harmonic solution is stored into the database using the **SET (p. 1724)** command (*KIMG* = AMPL); then the default is OFF.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

If Multi-Plots are not being displayed, and the current device is a 3D device (**/SHOW (p. 1785)**,3D), then the displacement scale in all active windows will be the same, even if separate **/DSCALE (p. 575)** commands are issued for each active window. For efficiency, the program maintains a single data structure (segment) containing only one displacement scale. The program displays the same segment (displacement scale) in all windows. Only the view settings will be different in each of the active windows.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Displacement Scaling

DSPOPTION,*Reord_Option*,*Memory_Option*,*Memory_Size*,--,--,*Solve_Info*

Sets memory option for the sparse solver.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Reord_Option

Reordering option:

DEFAULT

Use the default reordering scheme.

SEQORDER

Use a sequential equation reordering scheme. Relative to PARORDER, this option typically results in longer equation ordering times and therefore longer overall solver times. Occasionally, however, this option will produce better quality orderings which decrease the matrix factorization times and improve overall solver performance.

PARORDER

Use a parallel equation reordering scheme. Relative to SEQORDER, this option typically results in shorter equation ordering times and therefore shorter overall solver times. Occasionally, however, this option will produce lower quality orderings which increase the matrix factorization times and degrade overall solver performance.

Memory_Option

Memory allocation option:

DEFAULT

Use the default memory allocation strategy for the sparse solver. The default strategy attempts to run in the INCORE memory mode. If there is not enough physical memory available

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when the solver starts to run in the INCORE memory mode, the solver will then attempt to run in the OUTOFCORE memory mode.

INCORE

Use a memory allocation strategy in the sparse solver that will attempt to obtain enough memory to run with the entire factorized matrix in memory. This option uses the most amount of memory and should avoid doing any I/O. By avoiding I/O, this option achieves optimal solver performance. However, a significant amount of memory is required to run in this mode, and it is only recommended on machines with a large amount of memory. If the allocation for in-core memory fails, the solver will automatically revert to out-of-core memory mode.

OUTOFCORE

Use a memory allocation strategy in the sparse solver that will attempt to allocate only enough work space to factor each individual frontal matrix in memory, but will share the entire factorized matrix on disk. Typically, this memory mode results in poor performance due to the potential bottleneck caused by the I/O to the various files written by the solver.

FORCE

This option, when used in conjunction with the *Memory_Size* option, allows you to force the sparse solver to run with a specific amount of memory. This option is only recommended for the advanced user who understands sparse solver memory requirements for the problem being solved, understands the physical memory on the system, and wants to control the sparse solver memory usage.

Memory_Size

Initial memory size allocation for the sparse solver for each process (*Memory_Size*/process in GB). The *Memory_Size* setting should always be well within the physical memory available, but not so small as to cause the sparse solver to run out of memory. Warnings and/or errors from the sparse solver will appear if this value is set too low. If the FORCE memory option is used, this value is the amount of memory allocated for the entire duration of the sparse solver solution.

--,--

Unused fields

Solve_Info

Solver output option:

OFF

Turns off additional output printing from the sparse solver (default).

PERFORMANCE

Turns on additional output printing from the sparse solver, including a performance summary and a summary of file I/O for the sparse solver. Information on memory usage during assembly of the global matrix (that is, creation of the `Jobname.full` file) is also printed with this option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Automatic memory allocation is used.

Notes

This command controls options related to the sparse solver in all analysis types where this solver can be used.

The amount of memory required for the sparse solver is unknown until the matrix structure is preprocessed, including equation reordering. The amount of memory allocated for the sparse solver is then dynamically adjusted to supply the solver what it needs to compute the solution.

If you have a large memory system, you may want to try selecting the INCORE memory mode for larger jobs to improve performance. Also, when running the sparse solver with many processors on the same machine or on a machine with very slow I/O performance (for example, slow hard drive speed), you may want to try using the INCORE memory mode to achieve better performance. However, doing so may require much more memory compared to running in the OUTOFCORE memory mode.

Running with the INCORE memory mode is best for jobs which comfortably fit within the limits of the physical memory on a given system. If the sparse solver workspace exceeds physical memory size, the system will be forced to use virtual memory (or the system page/swap file). In this case, it is typically more efficient to run with the OUTOFCORE memory mode.

Menu Paths

This command cannot be accessed from a menu.

DSUM,*SIGNIF*,*Label*,*TD*,*ForceType*

Specifies the double sum mode combination method.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single-point, multi-point, or DDAM response spectrum (*Spectype* = SPRS, MPRS, or DDAM on **SPOPT (p. 1837)**), the significance level of a mode is defined as the mode coefficient divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT (p. 1837)**, PSD.)

Label

Label identifying the combined mode solution output.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

TD

Time duration for earthquake or shock spectrum. *TD* defaults to 10.

ForceType

Label identifying the forces to be combined:

STATIC

Combine the modal static forces (default).

TOTAL

Combine the modal static plus inertial forces.

Notes

Modes whose effective damping ratio defined in Equation 14.42 exceeds or equals 1.0 are not combined.

This command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine>DSUM Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine>DSUM Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine>DSUM Method

Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine>DSUM Method

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine>DSUM Method

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine>DSUM Method

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DSYM, *Lab*, *Normal*, *KCN*

Specifies symmetry or antisymmetry degree-of-freedom constraints on nodes.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Symmetry label:

SYMM

Generate symmetry constraints as described below (default).

ASYM

Generate antisymmetry constraints as described below.

Normal

Surface orientation label to determine the constraint set (surface is assumed to be perpendicular to this coordinate direction in coordinate system *KCN*):

X

Surface is normal to coordinate X direction (default). Interpreted as R direction for non-Cartesian coordinate systems.

Y

Surface is normal to coordinate Y direction. θ direction for non-Cartesian coordinate systems.

Z

Surface is normal to coordinate Z direction. Φ direction for spherical or toroidal coordinate systems.

KCN

Reference number of global or local coordinate system used to define surface orientation.

Notes

Specifies symmetry or antisymmetry degree-of-freedom constraints on the selected nodes. The nodes are first automatically rotated (any previously defined rotations on these nodes are redefined) into coordinate system *KCN*, then zero-valued constraints are generated, as described below, on the selected degree-of-freedom set (limited to displacement, velocity, and magnetic degrees of freedom) (**DOFSEL (p. 569)**). Constraints are defined in the (rotated) nodal coordinate system, as usual. See the **D (p. 483)** and **NROTAT (p. 1264)** commands for additional details about constraints and nodal rotations.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Symmetry and Antisymmetry Constraints:

Symmetry or antisymmetry constraint generations are based upon the valid degrees of freedom in the model, that is, the degrees of freedom associated with the elements attached to the nodes. The labels for degrees of freedom used in the generation depend on the *Normal* label.

For displacement degrees of freedom, the constraints generated are:

	SYMM		ASYM	
Normal	**2D**	**3D**	**2D**	**3D**
X	UX, ROTZ	UX, ROTZ, ROTY	UY	UY, UZ, ROTX
Y	UY, ROTZ	UY, ROTZ, ROTX	UX	UX, UZ, ROTY
Z	--	UZ, ROTX, ROTY	--	UX, UY, ROTZ

For velocity degrees of freedom, the constraints generated are:

	SYMM		ASYM	
Normal	**2D**	**3D**	**2D**	**3D**
X	VX	VX	VY	VY, VZ
Y	VY	VY	VX	VX, VZ
Z	--	VZ	--	VX, VY

For the 2D vector magnetic degree of freedom, AZ, symmetry is naturally satisfied and the SYMM label generates no constraints. The ASYM label generates flux parallel conditions (flux flows parallel to the surface).

	SYMM	ASYM
Normal	**2D**	**2D**
X	--	AZ
Y	--	AZ

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Normal>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Flux Par'l>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Antisymm B.C.>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Displacement>Symmetry B.C.>On Nodes

---

DSYS, *KCN*

Activates a display coordinate system for geometry listings and plots.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Coordinate system reference number. *KCN* may be 0,1,2 or any previously defined local coordinate system number.

Note:

If a cylinder is displayed in its cylindrical coordinate system (with a 1,0,0 view), it will be unrolled (developed) into a flat plane (with theta along the Y direction).

Command Default

Global Cartesian (*KCN* = 0) display coordinate system.

Notes

Boundary condition symbols, vector arrows, and element coordinate system triads are not transformed to the display coordinate system. The display system orientation (for the default view) is X horizontal to the right, Y vertical upward, and Z out of the screen (normal).

Line directions and area directions (**/PSYMB (p. 1552)**,LDIR and **/PSYMB (p. 1552)**,ADIR) are not plotted for *KCN* >0.

When you create 3D annotation, the coordinates are stored to the database in the display coordinate system that was active at the time of creation. Changing the display coordinate system does not change the annotation coordinate data in the database.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Change Display CS to>Global Cartesian

Utility Menu>WorkPlane>Change Display CS to>Global Cylindrical

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Utility Menu>WorkPlane>Change Display CS to>Global Spherical

Utility Menu>WorkPlane>Change Display CS to>Specified Coord Sys

---

DTRAN

Transfers solid model DOF constraints to the finite element model.

SOLUTION (p. 38): Solid Constraints (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Constraints are transferred only from selected solid model entities to selected nodes. The **DTRAN (p. 583)** operation is also done if the **SBCTRAN (p. 1647)** command is issued, and is automatically done upon initiation of the solution calculations (**SOLVE (p. 1822)**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Constraints

Main Menu>Solution>Define Loads>Operate>Transfer to FE>Constraints

---

DUMP, *NSTRT*, *NSTOP*

Dumps the contents of a binary file.

AUX2 (p. 61): Binary File Dump (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSTRT*, *NSTOP

Dump file from record *NSTRT* (defaults to 1) to *NSTOP* (defaults to *NSTRT*). If *NSTRT* = HEAD, dump only record 1 of the file (*NSTOP* and the format specification are ignored). If *NSTRT* = ALL, dump the entire file.

Notes

Dumps the file named on the AUX2 **FILEAUX2 (p. 753)** command according the format specified on the **FORM (p. 773)** command.

Menu Paths

Utility Menu>File>List>Binary Files

Utility Menu>List>Files>Binary Files

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/DV3D, *Lab*, *Key*

Sets 3D device option modes.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Mode label:

ACCU

Activating the accumulation buffer for OpenGL graphics, providing faster model rotation when shaded backgrounds are in use. This feature is off by default.

ACTR

Label term to designate the cursor position as the center for automatic dynamic rotational center capability. The subsequent *Key* value (see below) turns this capability on and off. This feature is on by default. (Available for OpenGL displays only.)

ANIM

Animation mode. The ANIM option allows you to create animation frames in pixmap mode instead of display list mode. This may improve large model performance, but it eliminates local manipulation while animation is in progress. This feature is on by default.

ANTI

Label term to control Anti-aliasing, a smoothing technique for your graph plots. (see below) The subsequent *Key* value turns this capability on and off. The default for this feature is off. (Available for OpenGL displays only).

CNTR

Switches banded contours on (1) or off (0) for your 3D contour display. The default is 1 (ON). Other contour parameters such as number of contours or the increment and range are defined using the **/CONTOUR (p. 415)** command. When either 9 or 128 contours are specified via **/CONTOUR (p. 415)**, this command is ignored and a smooth contour is always displayed.

DGEN

Local manipulation degenerate mode. You access the DGEN option to set wire-frame local manipulation mode for 3D devices (device dependent). This feature is off by default.

DLIST

With DLIST, you can specify whether screen updates and redraws will be performed using the Mechanical APDL display list (off), or the 3D device's display list (on). DLIST is on by default for Windows systems, but off for Linux.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELS

You use DELS to suppress contour display screen overwrites when **/NOERASE (p. 1249)** is active. This prevents the bleed-through that occurs when you overlay contour plots.

TRIS

Triangle strip mode. Tri-stripping provides faster 3D display capabilities and is on by default. Some display enhancements, such as texturing, are adversely affected by tri-stripping. You can turn off tri-stripping in order to improve these display functions. Be sure to turn tri-stripping on after the desired output is obtained.

Key

The following key options apply to *Lab* = ACCU:

0

(OFF) The accumulation buffer is not accessed. (default)

1

(ON) Access to the buffer is enabled.

The following key options apply to *Lab* = ACTR:

0

(OFF) The cursor position has no effect on the existing rotational center for dynamic operations.

1

(ON) The rotational center for dynamic rotations in OpenGL is determined by the position of the mouse cursor on (or within 15 pixels of) the model. Any rotations that are initiated with the cursor more than 15 pixels from the model will occur about the midpoint of the Z-axis at that point in space. If the Z-buffer has not been refreshed the Z-axis will have an infinite value, and rotations will appear to occur about an extremely long Z-axis. This behavior stops when the graphics window is refreshed or replotted. (default)

Note that when using the GUI in 3D mode, when ACTR = 1, the **Rotational Center** option is grayed out under **Utility Menu> PlotCtrls> View Setting** because the rotational center is determined strictly by the position of the mouse cursor.

The following key options apply to *Lab* = ANIM:

0

Display list animation. The object can be dynamically manipulated while animating. No legend, countour or annotation items are displayed. (see Notes, below)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

On Linux, device-dependent pixmap animation is used. On the PC, bitmap animation is provided (default). When you animate in this mode, you cannot dynamically manipulate your model (see Notes, below).

2

On the PC only, this option provides AVI animation which uses the AVI movie player .

Although you can create animations of multiple Mechanical APDL window schemes, animations created with OpenGL display lists (**/DV3D (p. 584)**, ANIM, 0) do not retain the windowing scheme information. You CAN save multiple windows via the X11/WIN32 drivers, or via the OpenGL driver with **/DV3D (p. 584)**, ANIM, KEY in effect (where KEY is not zero).

The following key options apply to *Lab* = ANTI

0

(OFF) Anti-aliasing is not active (default).

1

(ON) The anti-aliasing technique will be applied to smooth the lines in your displays (valid for OpenGL only).

The following key options apply to *Lab* = DGEN:

0

Normal manipulation.

1

Wireframe Manipulation.

The following key options apply to *Lab* = DLIST:

0

(OFF) The Mechanical APDL display list is used for plotting and dynamic graphics manipulation (Linux default).

1

(ON) The local (3D device) display list is used for plotting and dynamic rotation (Windows default).

The following key options apply to *Lab* = TRIS:

0

(OFF) Tri-stripping is off.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

(ON) Tri-stripping is on (default).

This command is valid in any processor.

Notes

Mechanical APDL uses display list animation for its 3D models. This memory resident array method interfaces with the OpenGL model information to allow the program to efficiently pan, zoom, rotate and dynamically manipulate your model during animation. The logo, legend, contour and other annotation items are produced in 2D and will not appear when **/DV3D (p. 584)**, ANIM, 0 is in effect. To display these items, use **/DV3D (p. 584)**, ANIM, 1. All screen data will be displayed, but manipulation of the model will not be possible.

Menu Paths

Utility Menu>PlotCtrls>Device Options

DVAL, *BaseID*, *Lab*, *VALUE*, *VALUE2*, *KeyCal*

Defines values at enforced motion base.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

BaseID

The identification number of the enforced motion base (defined using the **D (p. 483)** command in the modal analysis).

Lab

U

Enforced displacement.

ACC

Enforced acceleration.

VALUE

The value or table name reference for tabular boundary conditions. To specify a table, enclose the table name in percent (%) signs (**DVAL (p. 587)**,*BaseID*,U,%*tablename*%). Use the ***DIM (p. 530)** command to define a table.

VALUE2

The value of the second degree of freedom (if present). If the analysis type and the degree of freedom allow a complex input, *VALUE* is the real component and *VALUE2* is the imaginary component.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KeyCal

Displacement result calculation key:

ON

Calculate absolute displacement and acceleration results (default).

OFF

Calculate relative displacement and acceleration results.

Notes

In a mode-superposition harmonic or transient analysis, you can apply enforced displacement or acceleration loads. If multiple loads are specified for the same base identification number (BaseID), the last load applied overrides the previous ones. For example, the following commands apply displacement to the base with identification number 1:

```
DVAL,1,U,VALUE
DVAL,1,ACC,VALUE
```

In this case, the acceleration (ACC) applied in the last command will override the displacement (U).

Issue **LSCLEAR (p. 1071)**,LSOPT to delete **DVAL (p. 587)** command options from the database.

For more information, see Enforced Motion Method for Mode-Superposition Transient and Harmonic Analyses in the *Structural Analysis Guide* and Enforced Motion Method for Transient and Harmonic Analyses in the *Mechanical APDL Theory Reference*.

Menu Paths

This command cannot be accessed from a menu.

DVMORPH, *VOLU*, *XAREA*, *RMSHKY*

Move nodes in selected volumes to conform to structural displacements.

PREP7 (p. 22): Morphing (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VOLU

Non-structural volume to which mesh movement (morph) applies. If ALL, apply morphing to all selected volumes (**VSEL (p. 2163)**). If *VOLU* = P, graphical picking is enabled. A component may be substituted for *VOLU*.

XAREA

Areas to be excluded from morphing. If ALL, exclude all selected areas (**ASEL (p. 192)**). If *XAREA* = P, graphical picking is enabled. A component may be substituted for *XAREA*. If *XAREA* is blank

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(default), allow morphing of nodes attached to areas of the selected volumes (*VOLU*) which are not shared by unselected volumes. (See Notes for clarification).

RMSHKY

Remesh flag option:

0

Remesh the selected non-structural volumes only if mesh morphing fails.

1

Remesh the selected non-structural volumes and bypass mesh morphing.

2

Perform mesh morphing only and do not remesh.

Notes

The selected volumes should include only non-structural regions adjacent to structural regions. DVMORPH will morph the non-structural volumes to coincide with the deflections of the structural regions.

Nodes in the structural regions move in accordance with computed displacements. Displacements from a structural analysis must be in the database prior to issuing DVMORPH.

By default, nodes attached to areas can move along the areas. You can use *XAREA* to restrain nodes on certain areas.

By default (*RMSHKY* = 0), DVMORPH will remesh the selected non-structural volumes entirely if a satisfactory morphed mesh cannot be provided.

If boundary conditions and loads are applied directly to nodes and elements, the DVMORPH command requires that these be removed before remeshing can take place.

Exercise care with initial conditions defined by the **IC (p. 921)** command. Before a structural analysis is performed for a sequentially coupled analysis, the DVMORPH command requires that initial conditions be removed from all null element type nodes in the non-structural regions. Use **ICDELE (p. 923)** to delete the initial conditions.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Phys Morphing>Volumes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DYNOPT

Specifies "Dynamic analysis options" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Dynamics Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E Commands

E, I, J, K, L, M, N, O, P

Defines an element by node connectivity.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

I

Number of node assigned to first nodal position (node I). If *I* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

J, K, L, M, N, O, P

Number assigned to second (node J) through eighth (node P) nodal position, if any.

Notes

Defines an element by its nodes and attribute values. You can specify up to eight nodes. You can add more nodes if needed (**EMORE (p. 627)**).

The number of nodes required and the order in which they are specified are described in Element Input in the *Element Reference*. Elements are assigned a number automatically (**NUMSTR (p. 1301)**) as generated. The current (or default) MAT, TYPE, REAL, SECNUM and ESYS attribute values are also assigned to the element.

When creating elements with more than eight nodes (**E (p. 591)** followed by **EMORE (p. 627)**), it may be necessary disable shape-checking (**SHPP (p. 1787)**) before issuing this command.

If a valid element type can be created without using additional nodes (**EMORE (p. 627)**), this command creates that element. **EMORE (p. 627)** then modifies the element to include the additional nodes. If shape-checking is active, it occurs before **EMORE (p. 627)** is issued. If the shape-checking limits are exceeded, therefore, element creation may fail before **EMORE (p. 627)** modifies the element into an acceptable shape.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>Thru Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EALIVE, *ELEM*

Reactivates an element (for the birth and death capability).

SOLUTION (p. 38): Birth and Death (p. 47)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element to be reactivated. If ALL, reactivate all selected elements (**ESEL (p. 661)**). If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*. To specify a table, enclose the table name in percent signs (%), e.g. **EALIVE (p. 592)**,%*tabname*%.

Notes

Reactivates the specified element when the birth and death capability is being used. An element can be reactivated only *after* it has been deactivated (**EKILL (p. 616)**).

Reactivated elements have a zero strain (or thermal heat storage, etc.) state.

The usage of tabular input is described in Using TABLE Type Array Parameters to Control Birth and Death in the *Advanced Analysis Guide*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>Activate Elem

Main Menu>Solution>Load Step Opts>Other>Birth & Death>Activate Elem

ECPCHG

Optimizes degree-of-freedom usage in a coupled acoustic model.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

The **ECPCHG (p. 592)** command converts uncoupled acoustic element types to coupled acoustic element types that are attached to the fluid-structure interaction interface. Or it converts coupled acoustic element types to uncoupled acoustic element types that are not attached to the fluid-structure interaction interface. Issuing **ECPCHG (p. 592)** can dramatically reduce the size of the `Jobname.emat` file, compared to the model fully meshed with the coupled acoustic elements.

Performing the **ECPCHG (p. 592)** conversion on meshed volumes can create circumstances in which more than one element type is defined for a single volume.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the acoustic elements are coupled with shell elements (SHELL181 or SHELL281), you must set the fluid-structure interaction (FSI) flag by issuing the **SF (p. 1733)**,,FSI command before the **ECPCHG (p. 592)** command.

ECPCHG (p. 592) may add new element types to your model, or it may change the element type for existing acoustic elements. You should verify the defined element types with **ETLIST (p. 702)** and the element attributes with **ELIST (p. 619)** after using this command.

Menu Paths

This command cannot be accessed from a menu.

EDELE, *IEL1*, *IEL2*, *INC*

Deletes selected elements from the model.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IEL1*, *IEL2*, *INC

Delete elements from *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *INC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *INC* are ignored and all selected elements (**ESEL (p. 661)**) are deleted. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

Notes

Deleted elements are replaced by null or "blank" elements. Null elements are used only for retaining the element numbers so that the element numbering sequence for the rest of the model is not changed by deleting elements. Null elements may be removed (although this is not necessary) with the **NUMCMP (p. 1291)** command. If related element data (pressures, etc.) are also to be deleted, delete that data before deleting the elements. **EDELE (p. 593)** is for unattached elements only. You can use the **xCLEAR** family of commands to remove any attached elements from the database.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

Main Menu>Preprocessor>Modeling>Delete>Elements

Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/EDGE, *WN*, *KEY*, *ANGLE*

Displays only the common lines ("edges") of an object.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies. The default window is 1.

KEY

Edge key:

Elements Plots

0

Display common lines between all adjacent element faces.

1

Display only the common lines between non-coplanar faces (that is, show only the edges).

Contour Plots

0

Display only the common lines between non-coplanar faces.

1

Display common lines between all element faces.

ANGLE

Largest angle between two faces for which the faces are considered to be coplanar (0° to 180°). Defaults to 45°. A smaller angle produces more edges, a larger angle produces fewer edges.

Command Default

For element plots, display common lines between all adjacent element faces. For contour plots, display only the common lines between non-coplanar faces.

Notes

The *ANGLE* field is used in PowerGraphics to determine geometric discontinuities. It is a tolerance measure for the differences between the normals of the surfaces being considered. Values within the tolerance are accepted as coplanar (geometrically continuous). In postprocessing displays, results are not averaged across discontinuous surfaces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A surface can be displayed as an edge outline without interior detail. This is useful for both geometry and postprocessing displays. Element outlines are normally shown as solid lines for geometry and displacement displays. Lines common to adjacent "coplanar" element faces are removed from the display. Midside nodes of elements are ignored.

The **/SHRINK (p. 1794)** option is ignored with the edge option.

/EDGE (p. 594) is not supported for **PLESOL (p. 1401)** and **/ESHAPE (p. 665)** displays when in PowerGraphics mode (**/GRAPHICS (p. 866)**,POWER).

The **/EDGE (p. 594)** command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Edge Options

EEMBED

Generates bonded connections between intersecting elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

From selected elements, **EEMBED (p. 595)** identifies embedded elements that are partially or completely enclosed within other (base) elements, determines the interior intersection surfaces between the embedded and base elements, and generates special-purpose REINF265 reinforcing elements representing the bonded connection stiffness of the interior intersection surfaces.

Supported base elements are 3D structural solids (SOLID185, SOLID186, SOLID187, and SOLID285) and thermal solids (SOLID278, SOLID279, and SOLID291). Supported embedded elements are 3D structural links and beams (LINK180, BEAM188, and BEAM189) and thermal link (LINK33).

Before issuing **EEMBED (p. 595)**, define the base and embedded elements, then select the elements to be included for embedded connections. Mesh conformity between the base and embedded elements is not required. A combination of supported base and embedded element types is allowed.

The command has no arguments. Simply issue the command to perform the embedding procedure.

You can inspect newly created reinforcing element types, sections, and elements (**ETLIST (p. 702)**, **SLIST (p. 1795)**, **ELIST (p. 619)**, or **EPLOT (p. 648)**).

Do not issue other preprocessing commands (such as **ET (p. 686)**, **E (p. 591)**, **EMODIF (p. 626)**, and **SECTYPE (p. 1697)**) to create or modify the special-purpose reinforcing elements, element types, and sections.

Elements generated by **EEMBED (p. 595)** are not associated with the solid model.

Reinforcing elements do not account for subsequent modifications made to the base and embedded elements. To avoid inconsistencies, issue **EEMBED (p. 595)** only after the base elements are finalized. If you delete or modify base or embedded elements (for example, via **EDELE (p. 593)**, **EMODIF (p. 626)**,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ETCHG (p. 699), **EMID (p. 624)**, **EORIENT (p. 646)**, **NUMMRG (p. 1295)**, or **NUMCMP (p. 1291)**), remove all affected reinforcing elements and associated sections, and reissue **EEMBED (p. 595)**.

EEMBED (p. 595) creates new reinforcing sections (of *Subtype* = GCON (p. 1697)) containing details about the created REINF265 elements, then applies them to all newly generated special-purpose REINF265 elements. The number of new reinforcing sections depends on the number of new REINF265 elements. (You can examine the properties of new sections (**SLIST (p. 1795)**).) The program sets the ID number of the newest reinforcing section to the highest available section ID number in the model. After issuing **EEMBED (p. 595)**, the command shows the highest-numbered IDs (element type, element, and section). Do not overwrite a new reinforcing section when defining subsequent sections.

If performing a subsequent structural analysis after the thermal analysis, you can use **EEMBED (p. 595)** to convert the reinforcing elements for the structural analysis, as follows:

1. Convert the thermal base and embedded elements to the appropriate structural element (**ET (p. 686)** or **EMODIF (p. 626)**).
2. Select the reinforcing elements generated by **EEMBED (p. 595)** in the previous thermal analysis only.
3. Issue **EEMBED (p. 595)**.

   Result: The program modifies the attributes of the selected reinforcing elements so that they are compatible with the converted base and embedded elements.

Degree-of-freedom boundary conditions cannot be directly applied on embedded LINK/BEAM nodes.

For more information about using this command, see Direct-Embedding Workflow in the *Structural Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

EEXTRUDE, *Action*, *NELEM*, *SPACE*, *DIST*, *THETA*, *TFACT* , –, *BCKEY*

Extrudes 2D plane elements into 3D solids during a 2D to 3D analysis.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Specifies one of the following command behaviors:

AUTO

Extrudes plane elements (PLANE182 and PLANE183) based on the KEYOPT(3) setting (p. 598). Complementary elements are also extruded. (See Notes (p. 598) for more information.) This behavior is the default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLANE

Extrudes elements in the global Z direction. KEYOPT(3) of the parent plane elements is ignored.

AXIS

Extrudes elements about the global Y axis. KEYOPT(3) of the parent plane elements is ignored.

TANGENT

Extrudes plane and contact elements about the global Y axis. Target elements are extruded in the global Z direction.

TIRE

Extrudes plane and contact elements about the global Y axis in a 360-degree span. Target elements are extruded in the global Z direction *if outside the plane elements*. Mesh refinement is adapted specifically for tire analysis.

See Figure 2: Example Command Actions (p. 599).

NELEM

Number of elements to generate in the extruded direction. If you do not specify a number, the program calculates a number automatically based on the average element size and extrusion distance.

SPACE

Spacing ratio. If positive, this value is the nominal ratio of the last division size to the first division size (if > 1.0, sizes increase, if < 1.0, sizes decrease). If negative, |SPACE| is the nominal ratio of the center division size to the end division size. Default = 1.0 (uniform spacing).

DIST

Distance to extrude in the global Z direction for the plane strain case (*Action* = PLANE). The default is 1.

THETA

Angle (in degrees) depending on *Action*:

Action = AXIS – Ending angle to extrude about the global Y axis for the axisymmetric case. Default = 360. (The beginning angle is always 0 degrees.)

Action = TIRE – Span of the contact patch for mesh refinement. The program generates an abrupt mesh transition from fine to coarse. Default = 0.

TFACT

Factor for increasing the rigid target size. The size of the extruded rigid target elements is determined automatically based on the size of the contact elements. Default = 0.2.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--

Reserved for future use.

BCKEY

Controls the nodal orientation in the third direction and boundary-condition mapping (*Action* = AXIS or TIRE only):

0 --

All nodes are rotated to a local Cartesian coordinate system where X is the radial, Y axial and Z circumferential direction. All loads and displacements are mapped from the 2D model to the 3D model in the local coordinate system.

If applying rotation ROTY in axisymmetric cases with torsion on the 2D model, this value sets UZ = 0 at all corresponding 3D nodes.

This value is the default.

1 --

Only nodes with applied loads and/or displacements are rotated to a local Cartesian coordinate system where X is the radial, Y axial and Z circumferential direction. All loads are mapped to the 3D model and all applied displacements are reset to zero.

See Figure 3: Boundary Condition Key Behavior (p. 600).

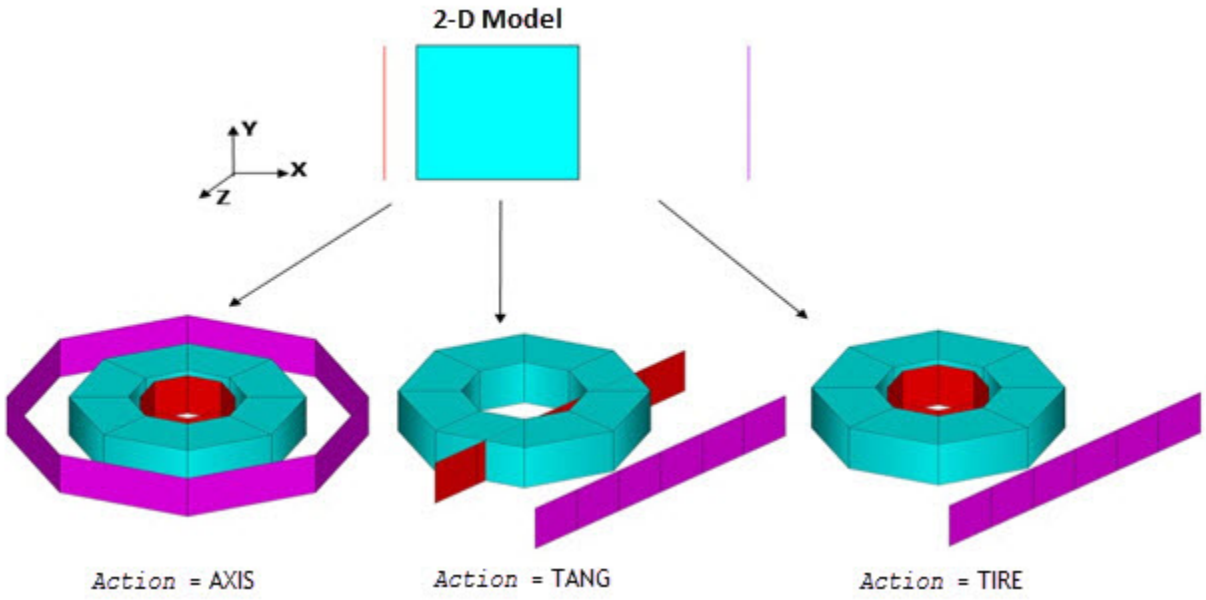
Notes

The **EEXTRUDE (p. 596)** command extrudes elements PLANE182 and PLANE183. Complementary elements TARGE169, CONTA172, and REINF263 also extrude. Extrusion operates automatically on elements in the selected element set.

Action = TIRE determines if target elements are in the middle (rim) part of the model or on the outside (road) part. The middle elements extrude axisymmetrically about the Y axis, and the outside elements extrude in the Z direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2: Example Command Actions



If interference exists between road and tire, the command extrudes outside elements within the specified tolerance (**SELTOL (p. 1710)**) in the global Z direction. For more information, see 2D to 3D analysis.

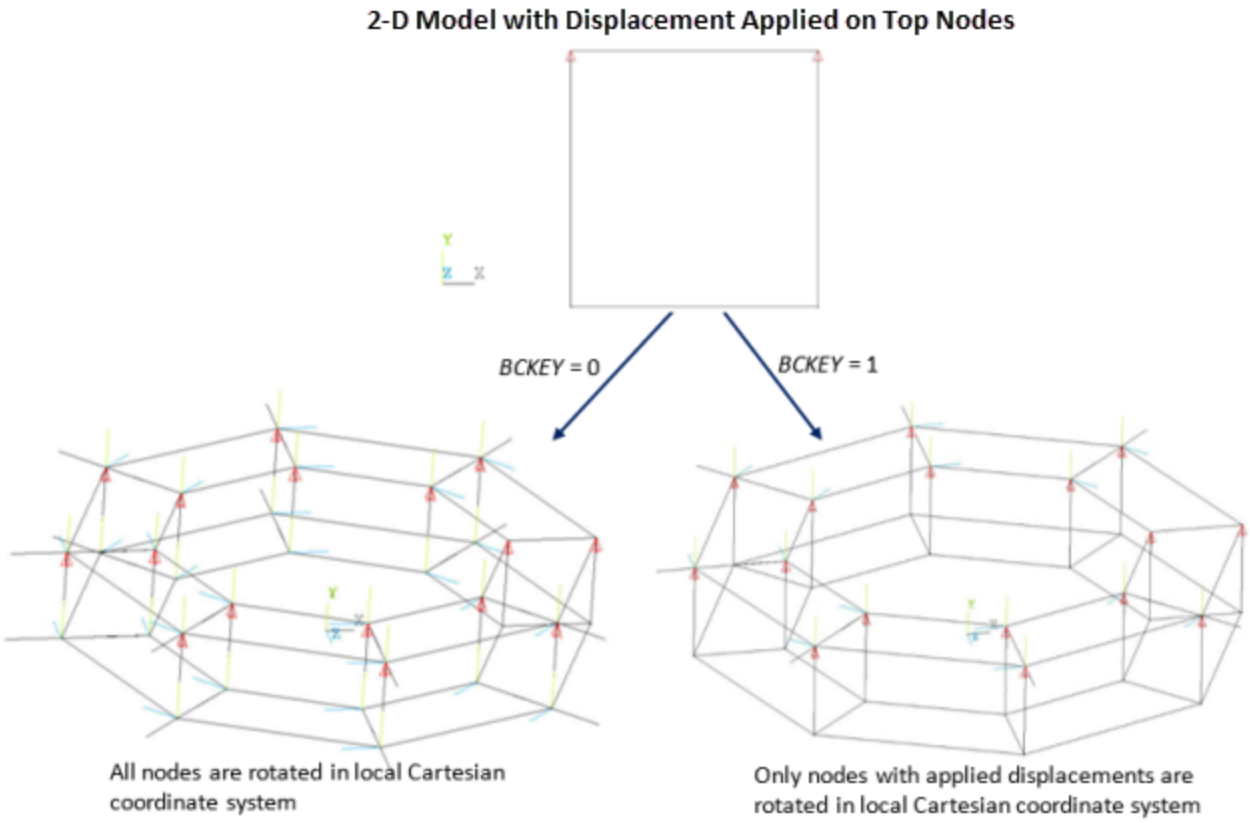
The *BCKEY* value is valid only within the 2D to 3D analysis environment (that is, after issuing **MAP2DTO3D (p. 1107)**,START and before issuing **MAP2DTO3D (p. 1107)**,FINISH).

Use the default *BCKEY* = 0 setting if you intend to apply minimal new loads or constraints during the 3D analysis phase; otherwise, set *BCKEY* = 1.

For more information, including how boundary conditions and loads are mapped from the 2D model to the 3D model, see 2D-to-3D Analysis in the *Advanced Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3: Boundary Condition Key Behavior



This command is valid in the PREP7 (**/PREP7 (p. 1479)**) and SOLUTION (**/SOLU (p. 1821)**) processors. Some options are valid within the 2D to 3D analysis environment only (between **MAP2DTO3D (p. 1107)**,START and **MAP2DTO3D (p. 1107)**,FINISH).

Element Behavior

For automatic PLANE182 and PLANE183 extrusion (*Action* = AUTO), based on the element behavior of the plane elements, the command performs as follows:

KEYOPT(3) = 0 --

Plane stress; the element is ignored.

KEYOPT(3) = 1 --

Axisymmetric; the element is extruded 360 degrees about the Y-axis. *THETA* is ignored.

KEYOPT(3) = 2 --

Plane strain (Z strain = 0.0); the element is extruded a unit distance in the global Z direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT(3) = 3 --

Plane stress with thickness input; the element is extruded in the Z-direction as specified by the thickness input via a real constant.

KEYOPT(3) = 5 --

Generalized plane strain; the element is ignored.

KEYOPT(3) = 6 --

Axisymmetric with torsion; the element is extruded 360 degrees about the Y-axis. *THETA* is ignored.

For an axisymmetric extrusion (*Action* = AUTO with KEYOPT(3) = 1, *Action* = AXIS, or *Action* = TANGENT), the command merges any nodes within the specified tolerance (**SELTOL (p. 1710)**,*TOLER*) of the axis into a single node, then forms degenerate tetrahedrons, pyramids, or wedges. The default tolerance value is 1.0E-6.

For an axisymmetric extrusion, SHELL208 and SHELL209 will extrude.

You can control shape-checking options via the **SHPP (p. 1787)** command.

The extrusion behavior of accompanying contact (CONTA172) is determined by the plane element settings. Rigid target (TARGE169) elements are extruded in the global Z direction unless axisymmetric extrusion (*Action* = AXIS or *Action* = TIRE) is in effect.

Within the 2D to 3D analysis environment (between **MAP2DTO3D (p. 1107)**,START and **MAP2DTO3D (p. 1107)**,FINISH), PLANE182, PLANE183, and associated contact/target/reinforcing elements are supported for the axisymmetric (with or without torsion) and plane-strain options only. For REINF263 reinforcing elements, if the fibers have an orientation angle that causes torsion in an axisymmetric analysis, use the axsiymmetrixc-with-torsion option (KEYOPT(3) = 6) for the base elements. For more information, see 2D-to-3D Analysis Requirements and Limitations in the *Advanced Analysis Guide*.

2D to 3D Element Pairs

The following table shows each 2D element capable of extrusion and its corresponding post-extrusion 3D element:

Pre-extrusion 2D Element	Post-extrusion 3D Element
PLANE182	SOLID185
PLANE183	SOLID186
TARGE169	TARGE170
CONTA172	CONTA174
REINF263	REINF265
SHELL208	SHELL181
SHELL209	SHELL281

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All element properties are also transferred consistently during extrusion. For example, a $\overline{B}$ 2D element is extruded to a $\overline{B}$ 3D element, and a mixed u-P 2D element is extruded to a mixed u-P 3D element. Element and node components are passed over the 3D elements and extruded nodes.

Menu Paths

This command cannot be accessed from a menu.

/EFACET, *NUM*

Specifies the number of facets per element edge for PowerGraphics displays.

POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUM

Number of facets per element edge for element plots.

1

Use 1 facet per edge (default for h-elements).

2

Use 2 facets per edge.

4

Use 4 facets per edge.

Command Default

As stated above.

Notes

/EFACET (p. 602) is valid only when PowerGraphics is enabled (**/GRAPHICS (p. 866)**,POWER), except that it can be used in FULL graphics mode for element CONTA174. (See the **/GRAPHICS (p. 866)** command and element CONTA174 in the *Element Reference* for more information.) The **/EFACET (p. 602)** command is only applicable to element type displays.

/EFACET (p. 602) controls the fineness of the subgrid that is used for element plots. The element is subdivided into smaller portions called *facets*. Facets are piecewise linear surface approximations of the actual element face. In their most general form, facets are warped planes in 3D space. A greater number of facets will result in a smoother representation of the element surface for element plots. **/EFACET (p. 602)** may affect results averaging. See Contour Displays in the *Basic Analysis Guide* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For midside node elements, use *NUM* = 2; if *NUM* = 1, no midside node information is output. For non-midside node elements, *NUM* should be set to 1. See the **PLNSOL (p. 1425)** and **PRNSOL (p. 1505)** commands for more information.

With PowerGraphics active (**/GRAPHICS (p. 866)**,POWER), the averaging scheme for surface data with interior element data included (**AVRES (p. 212)**,,FULL) and multiple facets per edge (**/EFACET (p. 602)**,2 or **/EFACET (p. 602)**,4) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE (p. 2034)**,,6 or **/TYPE (p. 2034)**,,7). When the Section data is not included in the averaging schemes (**/TYPE (p. 2034)**,,7), the resulting absolute value for the midside node is significantly smaller.

For cyclic symmetry mode-superposition harmonic solutions, only *NUM* = 1 is supported in postprocessing.

Caution:

If you specify **/EFACET (p. 602)**,1, *PowerGraphics does not plot midside nodes. You must use* **/EFACET (p. 602)**,2 *to make the nodes visible.*

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Options for Outp

Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu

Utility Menu>List>Results>Options

Utility Menu>Plot>Results>Contour Plot>Nodal Solution

Utility Menu>PlotCtrls>Multi-Plot Contrls

Utility Menu>PlotCtrls>Style>Size and Shape

EGEN, *ITIME*, *NINC*, *IEL1*, *IEL2*, *IEINC*, *MINC*, *TINC*, *RINC*, *CINC*, *SINC*, *DX*, *DY*, *DZ*

Generates elements from an existing pattern.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME*, *NINC

Do this generation operation a total of *ITIME*s, incrementing all nodes in the given pattern by *NINC* each time after the first. *ITIME* must be >1 if generation is to occur. *NINC* may be positive, zero, or negative. If *DX*, *DY*, and/or *DZ* is specified, *NINC* should be set so any existing nodes (as on **NGEN (p. 1218)**) are not overwritten.

IEL1*, *IEL2*, *IEINC

Generate elements from selected pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* is negative, *IEL2* and *IEINC* are ignored and the last |*IEL1*|

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elements (in sequence backward from the maximum element number) are used as the pattern to be repeated. If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and use all selected elements (**ESEL (p. 661)**) as pattern to be repeated. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

MINC

Increment material number of all elements in the given pattern by *MINC* each time after the first.

TINC

Increment element type number by *TINC*. The element types with incremented numbers must be defined before issuing the **EGEN (p. 603)** command.

RINC

Increment real constant table number by *RINC*.

CINC

Increment element coordinate system number by *CINC*.

SINC

Increment section ID number by *SINC*.

DX*, *DY*, *DZ

Define nodes that do not already exist but are needed by generated elements (as though the **NGEN (p. 1218)**,*ITIME*, *INC*, *NODE1*, , , *DX*, *DY*, *DZ* were issued before **EGEN (p. 603)**). Zero is a valid value. If blank, *DX*, *DY*, and *DZ* are ignored.

Notes

A pattern may consist of any number of previously defined elements. The MAT, TYPE, REAL, ESYS, and SECNUM numbers of the new elements are based upon the elements in the pattern and not upon the current specification settings.

You can use the **EGEN (p. 603)** command to generate interface elements (INTER192, INTER193, INTER194, and INTER195) directly. However, because interface elements require that the element connectivity be started from the bottom surface, you must make sure that you use the correct element node connectivity. See the element descriptions for INTER192, INTER193, INTER194, and INTER195 for the correct element node definition.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Elements>Auto Numbered

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EGID, *VAL*

Specifies a global identifier for a set of MESH200 elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL

An integer for identifying a set of MESH200 elements used in a reinforcing analysis. Default = 1.

Notes

VAL is a global identifier that you assign to a set of MESH200 elements in a reinforcing analysis. The command is valid only when using the mesh-independent method for defining reinforcing.

The global identifier that you specify is transferred to the reinforcing members (individual reinforcings) when they are generated (**EREINF (p. 656)**).

Issue **EGID (p. 605)** *before* generating the MESH200 elements.

If necessary, you can change the global identifier for an existing set of MESH200 elements (**EMODIF (p. 626)**).

For more information about using this command in a mesh-independent reinforcing analysis, see Selecting and Displaying Groups of Reinforcing Members in the *Structural Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

***EIGEN**, *Kmatrix*, *Mmatrix*, *Cmatrix*, *Evals*, *Evects*

Performs a modal solution with unsymmetric or damping matrices.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Kmatrix

Name of the stiffness matrix. May be a real or complex-valued matrix.

Mmatrix

Name of the mass matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cmatrix

Name of the damping matrix (used only for **MODOPT (p. 1140)**,DAMP).

Evals

Name of the output eigenvalues vector. It will be an *m*-long ***VEC (p. 2085)** vector of complex values, where *m* is the number of eigenvalues requested (**MODOPT (p. 1140)**).

Evects

Name of the output eigenvector matrix. It will be a *n* x *m* ***DMAT (p. 551)** (dense) matrix of complex values, where *n* is the size of the matrix and *m* is the number of eigenvalues requested (**MODOPT (p. 1140)**).

Notes

Use the command **ANTYPE (p. 162)**,MODAL and the **MODOPT (p. 1140)** command to specify the modal solution options. Only **MODOPT (p. 1140)**,DAMP, **MODOPT (p. 1140)**,UNSYM, **MODOPT (p. 1140)**,LANB, and **MODOPT (p. 1140)**,SUBSP are supported.

***EIGEN (p. 605)** with Block Lanczos (LANB) only supports sparse matrices.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

This command cannot be accessed from a menu.

EINFIN,*CompName*,*PNODE(NREF1)*,*NREF2*,*NREF3*,*MATID*

Generates structural infinite elements from selected nodes.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | PrepPost | **Solver** | AS add-on

CompName

Component name containing the reference nodes for calculating the position of *poles* for generating INFIN257 structural infinite elements.

The number of poles and their positions are based on the number of included nodes:

- If the component includes only one node, the node becomes the pole node. The pole node is typically located at or near the geometric center of the finite element domain if there is a single pole.
- If the component includes two (2D) or three nodes (3D), the program constructs an infinite line or area, then calculates the pole positions by drawing a perpendicular line from a selected node to the line or area.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the component includes more than two (2D) or three nodes (3D), the program uses the first two or three nodes calculate the position of poles.

PNODE* or *NREF1

Node number for the direct input of the pole node. (A parameter or parametric expression is also valid.) Specify this value when no *CompName* has been specified. (If *CompName* is specified, this value is ignored.)

If *NREF2*, or *NREF2* and *NREF3*, are defined, this value becomes the first reference node (*NREF1*) used to calculate pole positions.

NREF2

Node number of the second reference node used to calculate pole positions. (A parameter or parametric expression is also valid.) Specify this value when no *CompName* has been specified. (If *CompName* is specified, this value is ignored.) To input this value, first specify *NREF1* (replacing *PNODE*). Pole positions are calculated on an infinite line defined by *NREF1* and *NREF2*.

NREF3

Node number of the third reference node used to calculate pole positions. (A parameter or parametric expression is also valid.) Specify this value when no *CompName* has been specified. (If *CompName* is specified, this value is ignored.) To input this value, first specify *NREF1* (replacing *PNODE*) and *NREF2*. Pole positions are calculated on an infinite line defined by *NREF1*, *NREF2*, and *NREF3*.

MATID

Optional material ID of the structural infinite element. (A parameter or parametric expression is also valid.) If specified, this value defines the material properties of the structural infinite elements explicitly; otherwise, the material ID is copied from the base elements.

Notes

The **EINFIN (p. 606)** command generates structural infinite elements (INFIN257) directly from the selected face of valid base elements (existing standard elements in your model). The command scans all base elements for the selected nodes and generates a compatible infinite element type for each base element. A combination of different base element types is allowed if the types are all compatible with the infinite elements.

The infinite element type requires no predefinition (**ET (p. 686)**).

The faces of base elements are determined from the selected node set (**NSEL (p. 1266)**), and the geometry of the infinite element is determined based on the shape of the face. Element characteristics and options are determined according to the base element. For the face to be used, all nodes on the face of a base element must be selected

Use base elements to model the near-field domain that interacts with the solid structures or applied loads. To apply the truncated far-field effect, a single layer of infinite elements must be attached to the near-field domain. The outer surface of the near-field domain must be convex.

The material of the structural infinite elements can be defined in either of two ways. If *MATID* is specified, the command uses that value to define the material property of the structural infinite elements. If

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MATID is *not* specified, the material ID of the base element is copied to the attached infinite element. Although only isotropic material is allowed for the infinite elements, these options enable you to define anisotropic material for the base elements in a static analysis. (In a transient analysis, however, use the same isotropic material for both base elements and infinite elements.)

After the **EINFIN (p. 606)** command executes, you can verify the newly created infinite element types and elements (**ETLIST (p. 702)**, **ELIST (p. 619)**, **EPLOT (p. 648)**).

Infinite elements do not account for any subsequent modifications made to the base elements. It is good practice to issue the **EINFIN (p. 606)** command only *after* the base elements are finalized. If you delete or modify base elements (via **EDELE (p. 593)**, **EMODIF (p. 626)**, **ETCHG (p. 699)**, **EMID (p. 624)**, **EORIENT (p. 646)**, **NUMMRG (p. 1295)**, or **NUMCMP (p. 1291)** commands, for example) after generating the structural infinite elements, remove all affected infinite elements and reissue the **EINFIN (p. 606)** command; doing so prevents inconsistencies.

Command Usage Examples

Example 3: Single Reference Node (Pole)

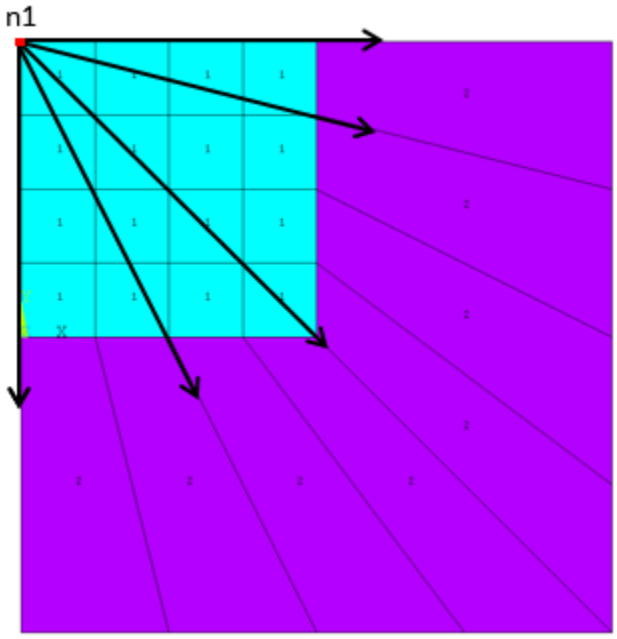
2D Elements:

```
/prep7
et,1,182
mp,ex,1,1e9
mp,nuxy,1,0.3

type,1             ! Generate solid elements
mat,1
rect,0,4,0,4
elsize,1
allsel
mshkey,1
amesh,1

n1 = node(0,4,0)   ! Select reference node
nsel,s,loc,x,4     ! Select nodes on base element
nsel,a,loc,y,0
EINFIN, ,n1
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3D Elements:

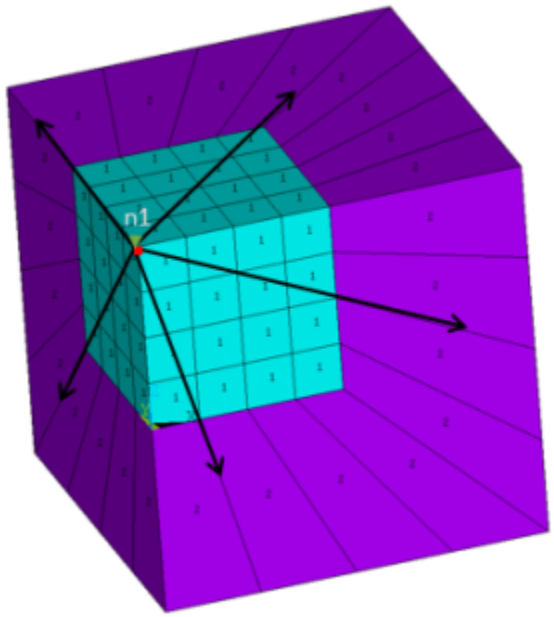
```
/prep7
et,1,185          ! Generate solid elements
mp,ex,1,1e9
mp,nuxy,1,0.3

type,1
mat,1
block,0,4,0,4,0,4
elsize,1

mshkey,1
vmesh,1

n1 = node(0,0,4)  ! Select reference node
nsel,,loc,x,4     ! Select nodes on base elements
nsel,a,loc,y,4
nsel,a,loc,z,0
EINFIN, ,n1
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Example 4: Two Reference Nodes

2D Elements:

```
/prep7
et,1,182            ! Generate solid elements
mp,ex,1,1e9
mp,nuxy,1,0.3
type,1
mat,1
rect,0,4,0,4
elsize,1
allsel
mshkey,1
amesh,1

n1 = node(0,0,0)    ! Select reference nodes
n2 = node(0,4,0)

nsel,,loc,x,4       ! Select nodes on base elements
EINFIN, ,n1,n2
```

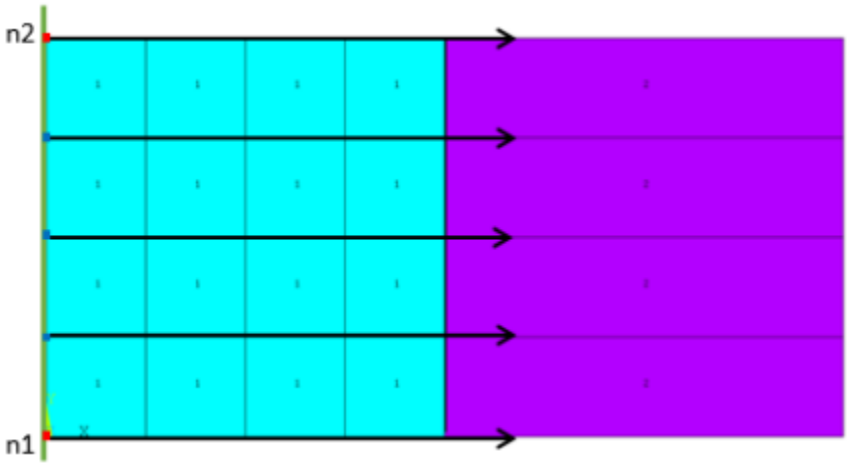
Following is the equivalent input with a component:

```
/prep7
et,1,182            ! Generate solid elements  mp,ex,1,1e9
mp,nuxy,1,0.3
type,1
mat,1
rect,0,4,0,4
elsize,1
allsel
mshkey,1
amesh,1

n1 = node(0,0,0)     ! Select  reference nodes
n2 = node(0,4,0)
nsel,,node,,n1
nsel,a,node,,n2
cm, nrefs, node      ! Define component
allsel
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
nsel,,loc,x,4        ! Select nodes on base elements
EINFIN, nrefs
```



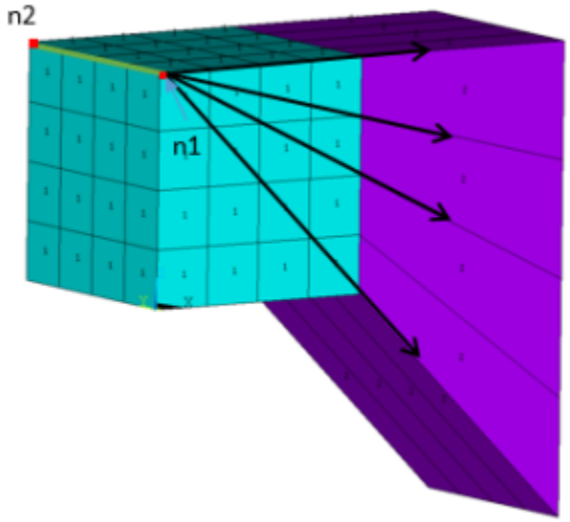
3D Elements:

```
/prep7

et,1,185             ! Generate solid elements
mp,ex,1,1e9
mp,nuxy,1,0.3

type,1
mat,1
block,0,4,0,4,0,4
elsize,1
mshkey,1
vmesh,1

n1 = node(0,0,4)    ! Select reference nodes
n2 = node(0,4,4)
nsel,,loc,x,4       ! Select nodes on base elements
EINFIN, ,n1, n2
```



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 5: Three Reference Nodes

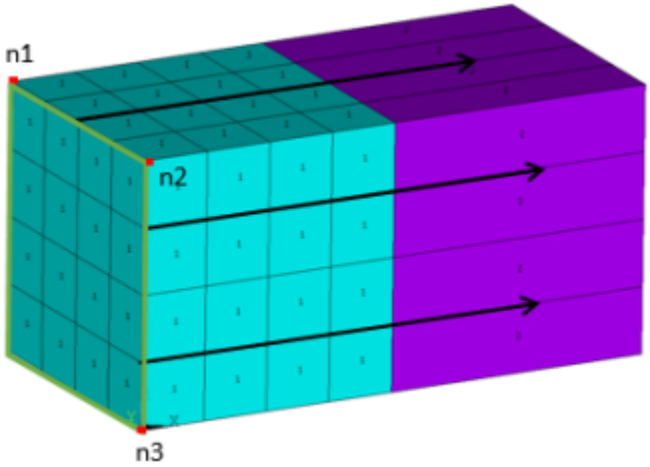
3D Elements Only:

```
/prep7

et,1,185            ! Generate solid elements
mp,ex,1,1e9
mp,nuxy,1,0.3

type,1
mat,1
block,0,4,0,4,0,4
elsize,1
mshkey,1
vmesh,1

n1 = node(0,4,4)    ! Select reference nodes
n2 = node(0,0,4)
n3 = node(0,0,0)
nsel,,loc,x,4       ! Select nodes on base elements
EINFIN, ,n1,n2,n3
```



Example 6: Multiple EINFIN Commands

2D Elements:

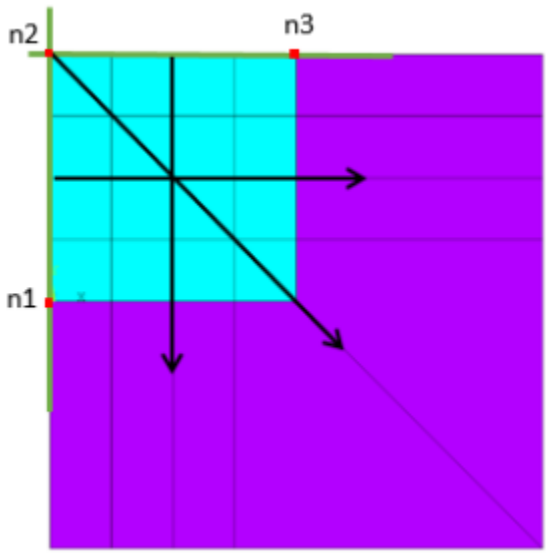
```
/prep7
et,1,182
mp,ex,1,1e9
mp,nuxy,1,0.3

type,1             ! Generate solid elements
mat,1
rect,0,4,0,4
elsize,1
allsel
mshkey,1
amesh,1

n1 = node(0,0,0)
n2 = node(0,4,0)
nsel,,loc,x,4,
nsel,r,loc,y,0.5,5
EINFIN,,n1,n2      ! upper right quad. elements
nsel,all
nsel,,loc,x,4
nsel,r,loc,y,0,1.5
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
EINFIN,,n2          ! Lower corner element
nsel,all
n3 = node(4,4,0)
nsel,s,loc,y,0
EINFIN,,n2,n3       ! Lower elements (y<0)
```



3D Elements:

```
/prep7

et,1,185              ! Generate solid elements
mp,ex,1,1e9
mp,nuxy,1,0.3

type,1
mat,1

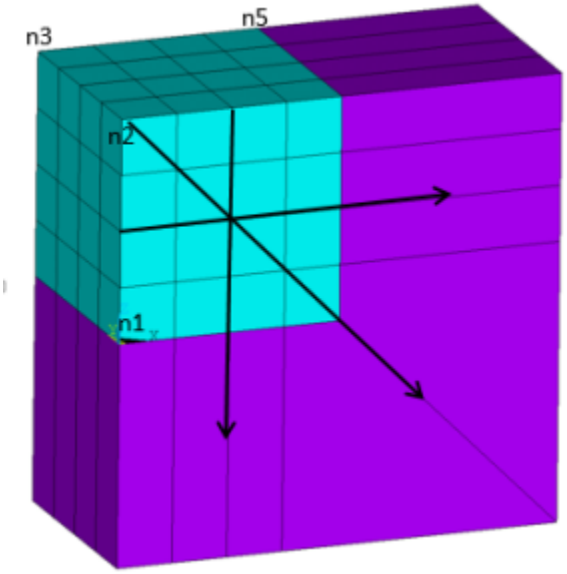
block,0,4,0,4,0,4
elsize,1

mshkey,1
vmesh,1

n1 = node(0,0,0)
n2 = node(0,0,4)
n3 = node(0,4,4)

nsel,s,loc,x,4,
nsel,u,loc,z,0
EINFIN, , n1, n2, n3  ! Upper right part
allsel
nsel,s,loc,x,4
nsel,r,loc,z,-0.1,1.2
EINFIN,,n2,n3         ! Corner element
allsel
n5 = node(4,4,4)
nsel,s,loc,z,0
EINFIN,,n2,n3,n5      ! Lower part (z<0)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Menu Paths

This command cannot be accessed from a menu.

EINTF, *TOLER*, *K*, *TLAB*, *KCN*, *DX*, *DY*, *DZ*, *KNONROT*

Defines two-node elements between coincident or offset nodes.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TOLER

Tolerance for coincidence (based on maximum Cartesian coordinate difference for node locations and on angle differences for node orientations). Defaults to 0.0001. Only nodes within the tolerance are considered to be coincident.

K

Only used when the type of the elements to be generated is PRETS179. *K* is the pretension node that is common to the pretension section that is being created. If *K* is not specified, it is created automatically and will have an Mechanical APDL-assigned node number. If *K* is specified but does not already exist, it will be created automatically but will have the user-specified node number. *K* cannot be connected to any existing element.

TLAB

Nodal number ordering. Allowable values are:

LOW

The 2-node elements are generated from the lowest numbered node to the highest numbered node.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HIGH

The 2-node elements are generated from the highest numbered node to the lowest numbered node.

REVE

Reverses the orientation of the selected 2-node element.

KCN

In coordinate system *KCN*, elements are created between node 1 and node 2 (= node 1 + dx dy dz).

DX, DY, DZ

Node location increments that define the node offset in the active coordinate system (DR, Dθ, DZ for cylindrical and DR, Dθ, DΦ for spherical or toroidal).

KNONROT

When *KNONROT* = 0, the nodes coordinate system is not rotated. When *KNONROT* = 1, the nodes belonging to the elements created are rotated into coordinate system *KCN* (see **NROTAT (p. 1264)** command description).

Notes

Defines 2-node elements (such as gap elements) between coincident or offset nodes (within a tolerance). May be used, for example, to "hook" together elements interfacing at a seam, where the seam consists of a series of node pairs. One element is generated for each set of two coincident nodes. For more than two coincident or offset nodes in a cluster, an element is generated from the lowest numbered node to each of the other nodes in the cluster. If fewer than all nodes are to be checked for coincidence, use the **NSEL (p. 1266)** command to select the nodes. Element numbers are incremented by one from the highest previous element number. The element type must be set (**ET (p. 686)**) to a 2-node element before issuing this command. Use the **CPINTF (p. 425)** command to connect nodes by coupling instead of by elements. Use the **CEINTF (p. 305)** command to connect the nodes by constraint equations instead of by elements.

For contact element CONTA178, the tolerance is based on the maximum Cartesian coordinate difference for node locations only. The angle differences for node orientations are not checked.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>At Coincid Nd

Main Menu>Preprocessor>Modeling>Create>Elements>Auto Numbered>Offset Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EKILL, *ELEM*

Deactivates an element (for the birth and death capability).

SOLUTION (p. 38): Birth and Death (p. 47)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element to be deactivated. If ALL, deactivate all selected elements (**ESEL (p. 661)**). If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *ELEM*. To specify a table, enclose the table name in percent signs (%), e.g. **EKILL (p. 616)**,%*tabname*%.

Notes

Deactivates the specified element when the birth and death capability is being used. A deactivated element remains in the model but contributes a near-zero stiffness (or conductivity, etc.) value (**ESTIF (p. 681)**) to the overall matrix. Any solution-dependent state variables (such as stress, plastic strain, creep strain, etc.) are set to zero. Deactivated elements contribute nothing to the overall mass (or capacitance, etc.) matrix, and do not generate a load vector (pressures, convections, gravity, etc.).

The usage of tabular input is described in Using TABLE Type Array Parameters to Control Birth and Death in the *Advanced Analysis Guide*.

The element can be reactivated with the **EALIVE (p. 592)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>Kill Elements

Main Menu>Solution>Load Step Opts>Other>Birth & Death>Kill Elements

ELBOW, *Transkey*, *TOL*, *Dof*, *Cons1*, *Cons2*, *Cons3*, *Cons4*

Specifies degrees of freedom to be coupled for end release and applies section constraints to elbow elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Transkey

Pipe-to-elbow transition flag:

OFF

Do not automatically transition pipes to elbows. (This behavior is the default.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON

Automatically convert straight PIPE289 elements to ELBOW290 elements where it is beneficial. The program converts elements in transition regions where curved ELBOW290 elements are connected to straight PIPE289 elements.

TOL

Angle tolerance (in degrees) between adjacent ELBOW290 elements. The default value is 20. A value of -1 specifies *all* selected ELBOW290 elements.

Dof

Degrees of freedom to couple:

ALL

Couple all nodal degrees of freedom (UX, UY, UZ, ROTX, ROTY, and ROTZ). This behavior is the default.

BALL

Create ball joints (equivalent to releasing ROTX, ROTY, and ROTZ).

Cons1*, *Cons2*, *Cons3*, *Cons4

Section degrees of freedoms to constrain. If *Cons1* through *Cons4* are unspecified, no section constraints are applied:

SECT

All section deformation

SE

Section radial expansion

SO

Section ovalization

SW

Section warping

SRA

Local shell normal rotation about cylindrical axis t2

SRT

Local shell normal rotation about cylindrical axis t1

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **ELBOW (p. 616)** command specifies end releases and section constraints for ELBOW290 elements and converts straight PIPE289 elements to ELBOW290 elements.

Curved PIPE289 elements are *not* converted to ELBOW290 elements.

ELBOW290 elements are generated only if there are *existing* ELBOW290 elements in the curved areas.

The command works on currently selected nodes and elements. It creates end releases on any two connected elbow elements whose angle at connection exceeds the specified tolerance. From within the GUI, the **Picked** node option generates an end release and section constraints at the selected node regardless of the angle of connection (that is, the angle tolerance [*TOL*] is set to -1).

Elbow and pipe elements must share the same section ID in order for the pipe-to-elbow transition to occur.

To list the elements altered by the **ELBOW (p. 616)** command, issue an **ELIST (p. 619)** command.

To list the coupled sets generated by the **ELBOW (p. 616)** command, issue a **CPLIST (p. 427)** command.

To list the section constraints generated by the **ELBOW (p. 616)** command, issue a **DLIST (p. 550)** command.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>End Releases>Elbows>On Selected set

Main Menu>Solution>Load Step Opts>Other>End Releases>Elbows>On Selected set

---

ELEM

Specifies "Elements" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ELIST, *IEL1*, *IEL2*, *INC*, *NNKEY*, *RKEY*

Lists the elements and their attributes.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IEL1*, *IEL2*, *INC

Lists elements from *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *INC* (defaults to 1). If *IEL1* = ALL (default), *IEL2* and *INC* are ignored and all selected elements (**ESEL (p. 661)**) are listed. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *INC* are ignored).

NNKEY

Node listing key:

0

List attribute references and nodes.

1

List attribute references but not nodes.

RKEY

Real constant listing key:

0

Do not show real constants for each element.

1

Show real constants for each element. This includes default values chosen for the element.

Notes

Lists the elements with their nodes and attributes (MAT, TYPE, REAL, ESYS, SECNUM, PART). See also the **LAYLIST (p. 1017)** command for listing layered elements.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Elements>Attributes + RealConst

Utility Menu>List>Elements>Attributes Only

Utility Menu>List>Elements>Nodes + Attributes

Utility Menu>List>Elements>Nodes + Attributes + RealConst

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*ELSE

Separates the final if-then-else block.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Optional final block separator within an if-then-else construct. See the ***IF (p. 927)** for details. If a batch input stream hits an end-of-file during a false ***IF (p. 927)** condition, the run will not terminate normally. You will need to terminate it externally (via either the Linux "kill" function or the Windows task manager). The ***ELSE (p. 620)** command must appear on the same file as the ***IF (p. 927)** command, and all five characters must be input.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

*ELSEIF, *VAL1*, *Oper1*, *VAL2*, *Conj*, *VAL3*, *Oper2*, *VAL4*

Separates an intermediate if-then-else block.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

VAL1

First numerical value (or parameter which evaluates to numerical value) in the conditional comparison operation. *VAL1*, *VAL2*, *VAL3*, and *VAL4* can also be character strings (enclosed in quotes) or parameters for *Oper* = EQ and NE only.

Oper1

Operation label. A tolerance of 1.0E-10 is used for comparisons between real numbers:

EQ --

Equal (for *VAL1* = *VAL2*).

NE --

Not equal (for *VAL1* ≠*VAL2*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LT --

Less than (for *VAL1*<*VAL2*).

GT --

Greater than (for *VAL1*>*VAL2*).

LE --

Less than or equal (for *VAL1* ≤ *VAL2*).

GE --

Greater than or equal (for *VAL1* ≥ *VAL2*).

ABLT --

Absolute values of *VAL1* and *VAL2* before < operation.

ABGT --

Absolute values of *VAL1* and *VAL2* before > operation.

VAL2

Second numerical value (or parameter which evaluates to numerical value) in the conditional comparison operation.

Conj

(Optional) Connection between two logical clauses.

AND -

True if both clauses (*Oper1* and *Oper2*) are true.

OR -

True if either clause is true.

XOR -

True if either (but not both) clause is true.

VAL3

(Optional) Third numerical value (or parameter which evaluates to numerical value).

Oper2

(Optional) Operation label. This will have the same labels as *Oper1*, except it uses *Val3* and *Val4*. A tolerance of 1.0E-10 is used for comparisons between real numbers.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL4

(Optional) Fourth numerical value (or parameter value which evaluates to a numerical value).

Notes

Optional intermediate block separator within an if-then-else construct. All seven characters of the command name (*ELSEIF) must be input. This command is similar to the ***IF (p. 927)** command except that the *Base* field is not used. The ***IF (p. 927)**, ***ELSEIF (p. 620)**, ***ELSE (p. 620)**, and ***ENDIF (p. 636)** commands for each if-then-else construct must all be read from the same file (or keyboard).

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

EMAGERR

Calculates the relative error in an electrostatic or electromagnetic field analysis.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

The relative error is an approximation of the mesh discretization error associated with a solution. It is based on the discrepancy between the unaveraged, element-nodal field values and the averaged, nodal field values. The calculation is valid within a material boundary and does not consider the error in continuity of fields across dissimilar materials.

For electrostatics, the field values evaluated are the electric field strength (EFSUM) and the electric flux density (DSUM). A relative error norm of each is calculated on a per-element basis and stored in the element table (**ETABLE (p. 687)**) with the labels EF_ERR and D_ERR. Normalized error values EFN_ERR and DN_ERR are also calculated and stored in the element table. Corresponding quantities for electromagnetics are H_ERR, B_ERR, HN_ERR, and BN_ERR, which are calculated from the magnetic field intensity (HSUM) and the magnetic flux density (BSUM). The normalized error value is the relative error norm value divided by the peak element-nodal field value for the currently selected elements.

Use the **PLETAB (p. 1411)** and **PRETAB (p. 1490)** commands to plot and list the error norms and normalized error values.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Error Eval

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMATWRITE, *Key*

Forces the writing of all the element matrices to `Jobname.emat`.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Write key:

YES

Forces the writing of the element matrices to `Jobname.emat` even if not normally done.

NO

Element matrices are written only if required. This value is the default.

Notes

The **EMATWRITE (p. 623)** command forces Mechanical APDL to write the `Jobname.emat` file.

If used in the solution processor (**/SOLU (p. 1821)**), this command is valid within the first load step only.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

EMF

Calculates the electromotive force (emf), or voltage drop along a predefined path.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

EMF (p. 623) invokes a Mechanical APDL macro which calculates the electromotive force (emf), or voltage drop along a predefined path (specified with the **PATH (p. 1354)** command). It is valid for both 2D and 3D electric field analysis. The calculated emf value is stored in the parameter EMF.

You must define a line path (via the **PATH (p. 1354)** command) before issuing the **EMF (p. 623)** command macro. The macro uses calculated values of the electric field (EF), and uses path operations for the calculations. All path items are cleared when the macro finishes executing.

The **EMF (p. 623)** macro sets the "ACCURATE" mapping method and "MAT" discontinuity option on the **PMAP (p. 1449)** command. The program retains these settings after executing the macro.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>EMF

EMFT

Summarizes electromagnetic forces and torques.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Use this command to summarize electromagnetic force and torque in both static electric and magnetic problems. To use this command, select the nodes in the region of interest and make sure that all elements are selected. If **RSYS (p. 1639)** = 0, the force is reported in the global Cartesian coordinate system. If **RSYS (p. 1639)** ≠ 0, force is reported in the specified coordinate system. However, for torque, if **RSYS (p. 1639)** ≠ 0, this command will account for the shift and rotation as specified by **RSYS (p. 1639)**, but will report only the Cartesian components.

Forces are stored as items _FXSUM, _FYSUM, _FZSUM, and _FSSUM. Torque is stored as items _TXSUM, _TYSUM, _TZSUM, and _TSSUM.

This command is valid only with PLANE121, SOLID122, SOLID123, PLANE222, PLANE223, SOLID225, SOLID226, SOLID227, PLANE233, SOLID236 and SOLID237 elements.

Menu Paths

Main Menu> General Postprocessor> Elec&Mag Calc> Summarize Force/Torque

---

EMID, *Key*, *Edges*

Adds or removes midside nodes.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Add or remove key:

ADD

Add midside node to elements (default).

REMOVE

Remove midside nodes from elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edges

ALL

Add (or remove) midside nodes to (from) all edges of all selected elements, independent of which nodes are selected (default).

EITHER

Add (or remove) midside nodes only to (from) element edges which have either corner node selected.

BOTH

Add (or remove) midside nodes only to (from) element edges which have both corner nodes selected.

Notes

This command adds midside nodes to (or removes midside nodes from) the selected elements. For this to occur, the selected elements must be midside node capable, the active element type (**TYPE (p. 2036)**) must allow midside node capability, and the relationship between the finite element model and the solid model (if any) must first be disassociated (**MODMSH (p. 1138)**).

By default, **EMID (p. 624)** generates a midside node wherever a zero (or missing) midside node occurs for that element. You can control this and add (or remove) midside nodes selectively by using the *Edges* argument. Nodes are located midway between the two appropriate corner nodes based on a linear Cartesian interpolation. Nodal coordinate system rotation angles are also linearly interpolated. Connected elements share the same midside node. Node numbers are generated sequentially from the maximum node number.

The **EMID (p. 624)** command is useful for transforming linear element types to quadratic element types having the same corner node connectivity.

EMID (p. 624) is also useful for transforming elements created outside of the program.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Add Mid Nodes

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Remove Mid Nd

---

EMIS,*MAT*,*EVALU*

Specifies emissivity as a material property for the Radiation Matrix method.

AUX12 (p. 62): Radiation Matrix Method (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Material number associated with this emissivity (500 maximum). Defaults to 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EVALU

Emissivity for this material (0.0 < *EVALU* ≤ 1.0). Enter a very small number for zero.

Command Default

Since there is no command default (p. 3) value for emissivity, you must issue **EMIS (p. 625)** to specify it. Otherwise, an error message appears. If you issue **EMIS (p. 625)** without defining a numerical value, emissivity defaults to 0.

Notes

Specifies emissivity as a material property for the Radiation Matrix method. This material property can then be associated with each element.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Emissivities

EMODIF, *IEL*, *STLOC*, *I1*, *I2*, *I3*, *I4*, *I5*, *I6*, *I7*, *I8*

Modifies a previously defined element.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IEL

Modify nodes and/or attributes for element number *IEL*. If ALL, modify all selected elements (**ESEL (p. 661)**). If *IEL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL*.

STLOC

Starting location (*n*) of first node to be modified or the attribute label.

If *n*, modify element node positions *n*, *n*+1, etc. (*n* = 1 to 20). For example, if *STLOC* = 1, *I1* refers to the first node, *I2*, the second, etc. If *STLOC* = 9, *I1* refers to the ninth node, *I2*, the tenth, etc. Attributes are also modified to the currently specified values.

Use -*n* to modify only nodes and not attributes.

If zero, modify only the attributes to the currently specified values.

If MAT, TYPE, REAL, ESYS, SECNUM, or EGID, modify only that attribute to the *I1* value.

I1, I2, I3, . . . , I8

Replace the previous node numbers assigned to this element with these corresponding values. A (blank) retains the previous value (except in the *I1* field, which resets the *STLOC* node number to zero).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For attributes (*STLOC* = MAT, TYPE, etc.), replace the existing value with the *I1* value (or the default if *I1* is zero or blank).

For attributes MAT and REAL, inputting the label GCN for *I1* replaces the existing attribute value with zero for the specified elements. This is a special case used only for converting contact elements (TARGE169 through CONTA177) from a pair-based definition to a general contact definition.

Notes

The nodes and/or attributes (MAT, TYPE, REAL, ESYS, SECNUM, or EGID values) of an existing element can be changed with this command.

The EGID attribute is valid only for MESH200 elements when they are used to generate reinforcing elements (REINF*nnn*) via the mesh-independent method for defining reinforcing.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Modify Attrib

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Modify Nodes

---

EMORE, *Q*, *R*, *S*, *T*, *U*, *V*, *W*, *X*

Adds more nodes to the just-defined element.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Q, R, S, T, U, V, W, X

Numbers of nodes typically assigned to ninth (node *Q*) through sixteenth (node *X*) nodal positions, if any. If *Q* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Repeat **EMORE (p. 627)** command for up to 4 additional nodes (20 maximum). Nodes are added after the last *nonzero* node of the element. Node numbers defined with this command may be zeroes.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMSEL, *Type*, - -, - -, *VMIN*, *VMAX*, *VINC*

Selects a group of reinforcing members via a predefined global identifier.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Specifies the selection type for the reinforcing elements:

S – Select a new set (default).

A – Select an additional set and add it to the current set.

U – Unselect a set from the current set.

ALL – Restore the full set.

STAT – Display the current selection status.

The following arguments are valid only when *Type* = S, A, or U:

- -

Reserved for future use.

- -

Reserved for future use.

VMIN

Minimum value of a group-identifier range.

VMAX

Maximum value of global identifier range. Default = VMIN for input values.

VINC

Value increment within the specified range. Default = 1.

Command Default

All global identifiers are selected.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Understanding Reinforcing Member Groups

When using the mesh-independent method for defining reinforcing, the global identifier for a set of MESH200 elements (specified via **EGID (p. 605)**) is transferred from the MESH200 elements to the reinforcing members (individual reinforcings) when the reinforcing elements (REINF*nnn*) are generated (**EREINF (p. 656)**).

The **EMSEL (p. 628)** command selects groups of reinforcing members (individual reinforcings) via specified global identifiers.

Example 7: Selecting Reinforcing Member Groups

The following command selects a new group of reinforcing members based on global identifiers 1 through 7:

```
EMSEL,S,,,1,7
```

VMIN, *VMAX*, and *VINC* are positive integer values.

This command is valid in PREP7 and POST1.

For more information about using this command in a mesh-independent reinforcing analysis, see Selecting and Displaying Groups of Reinforcing Members in the *Structural Analysis Guide*.

The **EMSEL (p. 628)** command is valid only for the reinforcing elements generated via the mesh-independent method for defining reinforcing.

When using the standard method for defining reinforcing, you can use the **LAYER (p. 1015)** command to select reinforcing members in POST1. (The **LAYER (p. 1015)** command is not valid in PREP7.)

Menu Paths

This command cannot be accessed from a menu.

EMSYM, *NSECT*

Specifies circular symmetry for electromagnetic sources.

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSECT

The number of circular symmetry sections (defaults to 1).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Specifies the number of times to repeat electromagnetic sources for circular symmetry. Applies to SOURC36 elements and to coupled-field elements with electric current conduction results in the database. Sources are assumed to be equally spaced over 360° about the global Cartesian Z axis.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Copy Sources

Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Copy Sources

---

EMTGEN,*Ncomp*,*Ecomp*,*PNcomp*,*DOF*,*GAP*,*GAPMIN*,*FKN*,*EPZERO*,*--*,*Smethod*

Generates a set of TRANS126 elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ncomp

Component name of the surface nodes of a structure which attach to the TRANS126 elements. The component name must be 32 characters or less, and you must enclose the name in single quotes in the **EMTGEN (p. 630)** command line.

Ecomp

Component name of the TRANS126 elements generated. The component name must be 32 characters or less, and you must enclose the name in single quotes in the **EMTGEN (p. 630)** command line. Defaults to EMTELM.

PNcomp

Component name of the plane nodes generated by the command at an offset (*GAP*) from the surface nodes. The component name must be 32 characters or less, and you must enclose the name in single quotes in the **EMTGEN (p. 630)** command line. Defaults to EMTPNO.

DOF

Active structural degree of freedom (DOF) for TRANS126 elements in the Cartesian coordinate system. You must enclose the DOF label in single quotes:

UX

Displacement in X direction.

UY

Displacement in Y direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UZ

Displacement in Z direction.

GAP

Initial gap distance from the surface nodes to the plane. Be sure to use the correct sign with respect to Ncomp node location.

GAPMIN

Minimum gap distance allowed (GAPMIN real constant) for TRANS126 elements. Defaults to the absolute value of (*GAP*)*0.05.

FKN

Contact stiffness factor used as a multiplier for a contact stiffness appropriate for bulk deformation. Defaults to 0.1. A negative value is interpreted as the modulus of elasticity on which the contact stiffness will be based.

EPZERO

Free-space permittivity. Defaults to 8.854e-6 (μMKS units).

--

Unused field.

Smethod

Stiffness method for TRANS126 elements (KEYOPT(6) setting). You must enclose the following labels in single quotes:

AUGM

Augmented stiffness method (default).

FULL

Full stiffness method. This method must be used in a linear perturbation harmonic analysis.

Notes

The **EMTGEN (p. 630)** command generates a set of TRANS126 elements between the surface nodes of a moveable structure and a plane of nodes, typically representing a ground plane. The plane of nodes is created by the command at a specified offset (*GAP*). Each element attaches to a surface node and to a corresponding node representing the plane. The generated plane nodes should be constrained appropriately for the analysis.

By default, the created elements use the augmented stiffness method (KEYOPT(6) = 1), which can help convergence. You can change to the full stiffness method (KEYOPT(6) = 0) by setting *Smethod* = FULL. The full stiffness method is required for a linear perturbation harmonic analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use TRANS126 elements for simulating fully coupled electrostatic structural coupling between a MEMS device and a plane, if the gap distance between the device and the plane is small compared to the overall surface area dimensions of the device. This assumption allows for a point-wise closed-form solution of capacitance between the surface nodes and the plane; i.e. CAP = EPZERO*AREA/GAP, where EPZERO if the free-space permittivity, AREA is the area associated with the node, and GAP is the gap between the node and the plane. The area for each node is computed using the ARNODE function. See the ***GET (p. 797)** command description for more information on the ARNODE function.

With a distributed set of TRANS126 elements attached directly to the structure and a plane (such as a ground plane), you can perform a full range of coupled electrostatic-structural simulations, including:

- Static analysis (due to a DC voltage or a mechanical load)
- Prestressed modal analysis (eigenfrequencies, including frequency-shift effects of a DC bias voltage)
- Prestressed harmonic analysis (system response to a small-signal AC excitation with a DC bias voltage or mechanical load)
- Large signal transient analysis (time-transient solution due to an arbitrary time-varying voltage or mechanical excitation)

The TRANS126 element also employs a node-to-node gap feature so you can perform contact-type simulations where the structure contacts a plane (such as a ground plane). The contact stiffness factor, FKN, is used to control contact penetration once contact is initiated. A smaller value provides for easier convergence, but with more penetration.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Transducers>Electromechanic>Node to Plane

---

EMUNIT, *Lab*, *VALUE*

Specifies the system of units for magnetic field problems.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Label specifying the type of units:

MKS

Rationalized MKS system of units (meters, amperes, henries, webers, etc.). Free-space permeability is set to 4 πe-7 henries/meter. Free-space permittivity is set to 8.85 e-12 F/m.

MUZRO

User defined system of units. Free-space permeability is set to the value input for *VALUE*. Other units must correspond to the permeability units. Relative permeability may be altered to absolute values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EPZRO

User defined system of units. Free-space permittivity is set to the value input for VALUE. Other units must correspond to the permittivity units.

VALUE

User value of free-space permeability (defaults to 1) if *Lab* = MUZRO, or free-space permittivity (defaults to 1) if *Lab* = EPZRO.

Command Default

Rationalized MKS system of units (meters, amperes, henries, webers, etc.). Free-space permeability is set to 4 πe-7 Henries/meter, free-space permittivity is set to 8.85e-12 Farads/meter.

Notes

Specifies the system of units to be used for electric and magnetic field problems. The free-space permeability and permittivity values may be set as desired. These values are used with the relative property values (**MP (p. 1160)**) to establish absolute property values.

Note:

If the magnetic source field strength (H_s) has already been calculated (**BIOT (p. 265)**), switching **EMUNIT (p. 632)** will not change the values.

For micro-electromechanical systems (MEMS), where dimensions are on the order of microns, see the conversion factors in System of Units in the *Coupled-Field Analysis Guide*.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Electromag Units

Main Menu>Preprocessor>Material Props>Electromag Units

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Electromag Units

---

EN, *IEL*, *I*, *J*, *K*, *L*, *M*, *N*, *O*, *P*

Defines an element by its number and node connectivity.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IEL

Number assigned to element being defined. If *IEL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I

Number of node assigned to first nodal position (node I).

J, K, L, M, N, O, P

Number assigned to second (node J) through eighth (node P) nodal position, if any.

Notes

Defines an element by its nodes and attribute values. Similar to the **E (p. 591)** command except it allows the element number (*IEL*) to be defined explicitly. Element numbers need not be consecutive. Any existing element already having this number will be redefined.

Up to 8 nodes may be specified with the **EN (p. 633)** command. If more nodes are needed for the element, use the **EMORE (p. 627)** command. The number of nodes required and the order in which they should be specified are described in the *Element Reference* for each element type. The current (or default) MAT, TYPE, REAL, SECNUM, and ESYS attribute values are also assigned to the element.

When creating elements with more than 8 nodes using this command and the **EMORE (p. 627)** command, it may be necessary to turn off shape checking using the **SHPP (p. 1787)** command before issuing this command. If a valid element type can be created without using the additional nodes on the **EMORE (p. 627)** command, this command will create that element. The **EMORE (p. 627)** command will then modify the element to include the additional nodes. If shape checking is active, it will be performed before the **EMORE (p. 627)** command is issued. Therefore, if the shape checking limits are exceeded, element creation may fail before the **EMORE (p. 627)** command modifies the element into an acceptable shape.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>User Numbered>Thru Nodes

---

/**ENCRYPT**,*Key*,*Fname*,*Ext*

Controls encryption of command input.

APDL (p. 19): Encryption/Decryption (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Encryption key used to encrypt the data (32-character maximum). A character parameter may be used. If the key is unspecified, encryption is turned off.

Fname

Name of file (including directory path) where the encrypted commands are written (248-character maximum for both file name and directory). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ext

File name extension (eight-character maximum).

Notes

This command opens the encrypted file specified by *Fname* and *Ext* for writing encrypted input commands.

Issuing this command results in a new file that overwrites any data in an existing file by the same name. When the encrypted file is written, the first line in the file is /DECRYPT,PASSWORD,OPENSSL and the last line is /DECRYPT.

See Encrypting Command Input and Other Data in the *Ansys Parametric Design Language Guide* for more information about using **/ENCRYPT (p. 634)** and performing encryption and decryption.

Menu Paths

This command cannot be accessed from a menu.

---

*END

Closes a macro file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Closes a file opened with ***CREATE (p. 434)**. The ***END (p. 635)** command is an 8-character command (to differentiate it from ***ENDIF (p. 636)**). If you add commented text on that same line but do not allow enough spaces between ***END (p. 635)** and the "!" that indicates the comment text, the ***END (p. 635)** will attempt to interpret the "!" as the 8th character and will fail.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*ENDDO

Ends a do-loop and starts the looping action.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

One ***ENDDO (p. 636)** is required for each nested do-loop. The ***ENDDO (p. 636)** command must appear on the same file as the ***DO (p. 567)** command, and all six characters must be input.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

*ENDIF

Ends an if-then-else.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Required terminator for the if-then-else construct. See ***IF (p. 927)** for details.

If a batch input stream hits an end-of-file during a false ***IF (p. 927)** condition, the analysis will not terminate normally. You will need to terminate it externally (use either the Linux "kill" function or the Windows task manager). The ***ENDIF (p. 636)** command must appear on the same file as the ***IF (p. 927)** command, and all six characters must be input.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ENDRELEASE, - -, *TOLERANCE*, *Dof1*, *Dof2*, *Dof3*, *Dof4*, *KJCT*, *KCHECK*

Specifies degrees of freedom to be decoupled for end release.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

- -

Unused field.

TOLERANCE

Angle tolerance (in degrees) between adjacent elements (*KCHECK* = 0), *or* a percentage tolerance value for comparing cross-section properties between adjacent elements (*KCHECK* = 1). Default = 20.

To release *all* selected elements, set *TOLERANCE* = -1.

See "Notes" (p. 638) for information about using this argument with *KCHECK*.

Dof1*, *Dof2*, *Dof3*, *Dof4

Degrees of freedom to release. If *Dof1* is blank, WARP is assumed and *Dof2*, *Dof3*, and *Dof4* are ignored.

WARP

Release the warping degree of freedom (default).

ROTX

Release rotations in the X direction.

ROTY

Release rotations in the Y direction.

ROTZ

Release rotations in the Z direction.

UX

Release displacements in the X direction.

UY

Release displacements in the Y direction.

UZ

Release displacements in the Z direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BALL

Create ball joints (equivalent to releasing WARP, ROTX, ROTY, and ROTZ).

KJCT

Behavior at a junction node (a node shared by more than two elements):

0 (or blank) – Release all elements. *TOLERANCE* is ignored.

1 – Release noncontinuous elements (if only one pair of continuous elements exists).

This argument is ignored (due to ambiguity) if *KJCT* = 1 and multiple pairs of continuous elements exist.

This argument is ignored if *TOLERANCE* = -1.

KCHECK

Controls how connected elements are checked at selected nodes.

0 (or blank) – Check only the angle between connected elements at the selected nodes (default).

1 – Check the angle and other cross-section properties (including offsets and orientations) between connected elements at the selected nodes.

This argument is ignored if *TOLERANCE* = -1.

Notes

ENDRELEASE (p. 637) specifies end releases for the BEAM188, BEAM189, PIPE288, and PIPE289 elements. The command works on currently selected nodes and elements.

Depending on the specified *KCHECK*, the command generates end releases on any two connected elements whose angle at the connection is > *TOLERANCE*, or whose cross-section properties have a difference > *TOLERANCE* percent.

From within the GUI, the **Picked node** option (equivalent to *TOLERANCE* = -1) generates an end release at the selected node regardless of the angle of connection or cross-section properties.

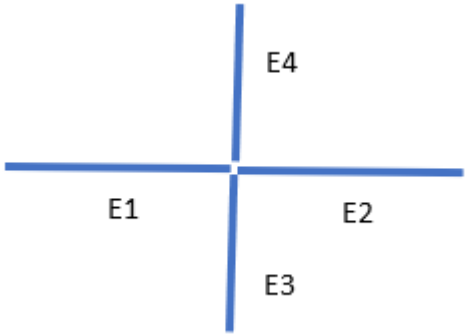
When *KCHECK* = 1:

- The specified *TOLERANCE* serves as a *percentage* for comparing cross-section properties between adjacent elements. (For example, if using the default *TOLERANCE* = 20, the tolerance is considered to be 20 percent.)
- In addition to the angle between connected elements, the section integrated properties (such as area, Iyy, Iyz, Izz, warping constant, and torsion constant), offsets, and cross-section orientations are also compared at the selected nodes. If the difference in any of the properties (or angles) is > *TOLERANCE*, the requested degrees of freedom are released for those nodes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For comparing cross-section orientations, the Y axes of cross-sections are compared after being projected onto the middle plane (the plane having a normal vector equal to the average of element X axes).
- For comparing offsets, the differences in the centers of cross-sections are compared against the approximate maximum section size of the connected elements.

Table 108: Examples: End-Release Conditions at a Junction Node when *KJCT* = 1 and *KCHECK* = 0

Example Junction Node (E – Element)	Behavior
	E2 is not released. E3 is released. A new node is generated for E3.
	E2 and E3 are released if the angles between E1-E2 and E1-E3 are within tolerance (two continuous pairs).
	E2, E3, and E4 are released regardless of *KJCT* (two continuous pairs).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

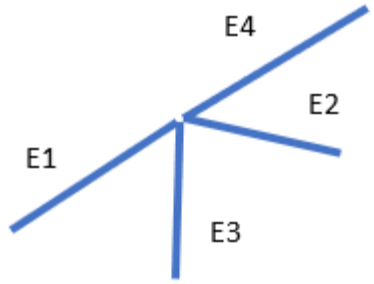
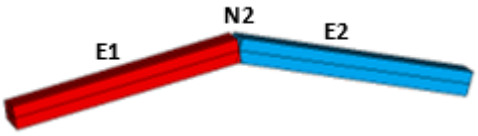
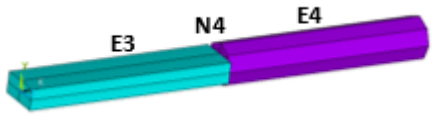
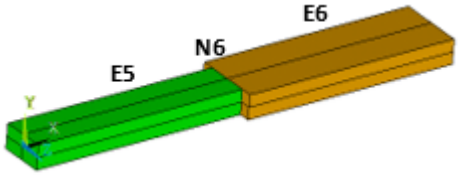
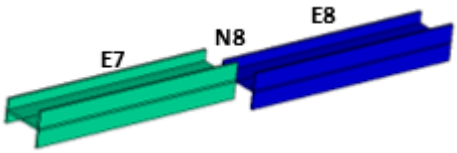
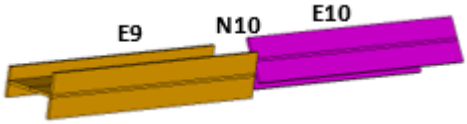
Example Junction Node (E – Element)	Behavior
	E4 is not released. E2 and E3 are released. New nodes are generated for E2 and E3.

Table 109: Examples: Using *KCHECK* with *TOLERANCE* = 20

Example (E – Element, N – Node)	Behavior when *KCHECK* = 0	Behavior when *KCHECK* = 1
	E1 and E2 are connected at an angle > 20°, so an end release is generated at N2.	E1 and E2 have the same section properties but are connected at an angle > 20°, so an end release is generated at N2.
	E3 and E4 are continuous (angle between them is < 20°), so an end release is not generated at N4.	E3 and E4 use different beam sections, so an end release is generated at N4.
	E5 and E6 are continuous (angle between them is less than 20°), so an end release is not generated at N6.	The difference between the section properties of E5 and E6 is > 20%, so an end release is generated at N6.
	E7 and E8 are continuous (angle between them are < 20°), so an end release is not generated at N8.	The difference between the section offsets of E7 and E8 is > 20%, so an end release is generated at N8.
	E9 and E10 are continuous (angle between them is < 20°), so an end release is not generated at N10.	The difference between the cross-section orientations of E9 and E10 is > 20°, so an end release is generated at N10.

To list the coupled sets generated by this command, issue **CPLIST (p. 427)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exercise engineering judgement when using this command. Improper use may result in mechanics that render a solution impossible.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>End Releases>Beams>On Selected set

Main Menu>Solution>Load Step Opts>Other>End Releases>Beams>On Selected set

---

ENERSOL, *NVAR*, *Item*, *--*, *Name*

Specifies the total energies to be stored.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number assigned to this variable (2 to NV).

Item

SENE

Potential energy (stiffness energy)

KENE

Kinetic energy

DENE

Damping energy

WEXT

Work done by external load

AENE

Artificial energy due to hourglass control/drill stiffness or due to contact stabilization damping

STEN

Artificial energy due to nonlinear stabilization

--

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NAME

A 32-character name identifying the item on printouts and displays. Defaults to a 4-character label formed by the four characters of the *Item* value.

Notes

Damping energy (DENE) and work done by external loads (WEXT) are available only if the following were set prior to the analysis solution: *EngCalc* = YES on the **TRNOPT (p. 2017)**, **HROUT (p. 918)** or **MXPAND (p. 1203)** command; and *Item* = VENG, ESOL, or ALL on the **OUTRES (p. 1336)** command.

If *EngCalc* = YES on the **HROUT (p. 918)** or **MXPAND (p. 1203)** command, *Item* = SENE and KENE are the average potential and kinetic energies, respectively.

Menu Paths

This command cannot be accessed from a menu.

ENGEN, *IINC*, *ITIME*, *NINC*, *IEL1*, *IEL2*, *IEINC*, *MINC*, *TINC*, *RINC*, *CINC*, *SINC*, *DX*, *DY*, *DZ*

Generates elements from an existing pattern.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IINC

Increment to be added to element numbers in pattern.

ITIME*, *NINC

Do this generation operation a total of *ITIME*s, incrementing all nodes in the given pattern by *NINC* each time after the first. *ITIME* must be > 1 if generation is to occur. *NINC* may be positive, zero, or negative.

IEL1*, *IEL2*, *IEINC

Generate elements from the pattern that begins with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* is negative, *IEL2* and *IEINC* are ignored and use the last |*IEL1*| elements (in sequence backward from the maximum element number) as the pattern to be repeated. If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and all selected elements (**ESEL (p. 661)**) are used as the pattern to be repeated. If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

MINC

Increment material number of all elements in the given pattern by *MINC* each time after the first.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TINC

Increment type number by *TINC*.

RINC

Increment real constant table number by *RINC*.

CINC

Increment element coordinate system number by *CINC*.

SINC

Increment section ID number by *SINC*.

DX*, *DY*, *DZ

Define nodes that do not already exist but are needed by generated elements (**NGEN (p. 1218)**,*ITIME*, *INC*, *NODE1*, , , *DX*, *DY*, *DZ*). Zero is a valid value. If blank, *DX*, *DY*, and *DZ* are ignored.

Notes

Same as the **EGEN (p. 603)** command except it allows element numbers to be explicitly incremented (*IINC*) from the generated set. Any existing elements already having these numbers will be redefined.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Elements>User Numbered

ENORM, *ENUM*

Reorients shell element normals or line element node connectivity.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ENUM

Element number having the normal direction that the reoriented elements are to match. If *ENUM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Reorients shell elements so that their outward normals are consistent with that of a specified element. **ENORM (p. 643)** can also be used to reorder nodal connectivity of line elements so that their nodal ordering is consistent with that of a specified element.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For shell elements, the operation reorients the element by reversing and shifting the node connectivity pattern. For example, for a 4-node shell element, the nodes in positions I, J, K and L of the original element are placed in positions J, I, L and K of the reoriented element. All 3D shell elements in the selected set are considered for reorientation, and no element is reoriented more than once during the operation. Only shell elements adjacent to the lateral (side) faces are considered.

The command reorients the shell element normals on the same panel as the specified shell element. A *panel* is the geometry defined by a subset of shell elements bounded by free edges or T-junctions (anywhere three or more shell edges share common nodes).

Reorientation progresses within the selected set until either of the following conditions is true:

- The edge of the model is reached.
- More than two elements (whether selected or unselected) are adjacent to a lateral face.

In situations where unselected elements might undesirably cause case b to control, consider using **ENSYM (p. 644)**,0,,0,ALL instead of **ENORM (p. 643)**. It is recommended that reoriented elements be displayed and graphically reviewed.

You cannot use the **ENORM (p. 643)** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constant values are not reoriented and may be invalidated by an element reversal.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Shell Normals

---

ENSYM, *IINC*, --, *NINC*, *IEL1*, *IEL2*, *IEINC*

Generates elements by symmetry reflection.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IINC

Increment to be added to element numbers in existing set.

--

Unused field.

NINC

Increment nodes in the given pattern by *NINC*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IEL1*, *IEL2*, *IEINC

Reflect elements from pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and pattern is all selected elements (**ESEL (p. 661)**). If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

Notes

The **ENSYM (p. 644)** command is similar to the **ESYM (p. 684)** command except that it enables explicitly assigned element numbers to the generated set (in terms of an increment *IINC*). Any existing elements already having these numbers are redefined.

The operation generates a new element by incrementing the nodes on the original element, and reversing and shifting the node connectivity pattern. For example, for a 4-node 2D element, the nodes in positions I, J, K and L of the original element are placed in positions J, I, L and K of the reflected element.

Similar permutations occur for all other element types. For line elements, the nodes in positions I and J of the original element are placed in positions J and I of the reflected element.

See the **ESYM (p. 684)** command for additional information about symmetry elements.

This command also provides a convenient way to reverse shell element normals. If the *IINC* and *NINC* argument fields are left blank, the effect of the reflection is to reverse the direction of the outward normal of the specified elements. You cannot use this command to change the normal direction of any element that has a body or surface load. It is best to apply all loading only *after* ensuring that the element normal directions are acceptable.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

For more information about controlling element normals, see Revising Your Model.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Shell Elems

Main Menu>Preprocessor>Modeling>Reflect>Elements>User Numbered

---

/EOF

Exits the file being read.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command causes an end-of-file exit when encountered on a switched file. (See **/INPUT (p. 948)**, ***USE (p. 2059)**, etc.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Commands are then read continuing from the file that contained the file switching command (or from the terminal if the switch was made from the terminal).

Use the **/EXIT (p. 708)** command to terminate a Mechanical APDL run.

This command cannot be used in a do-loop or if-then-else construct.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

EORIENT,*Etype*,*Dir*,*TOLER*

Reorients solid element normals.

PREP7 (p. 22): Meshing (p. 29)
PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Etype

Specifies which elements to orient.

LYSL

Specifies that layered solid elements will be oriented. This value is the default. Layered element types affected by this option include SOLID185 with KEYOPT(3) = 1, SOLID186 with KEYOPT(3) = 1, SOLSH190, SOLID278 with KEYOPT(3) = 1 or 2, and SOLID279 with KEYOPT(3) = 1 or 2.

Dir

The axis and direction for orientation, or an element number. If *Dir* is set to a positive number (*n*), then all eligible elements are oriented as similarly as possible to element *n*.

NEGX

The element face with the outward normal most nearly parallel to the element coordinate system's negative x-axis is designated (reoriented) as face 1.

POSX

The element face with the outward normal most nearly parallel to the element coordinate system's positive x-axis is designated (reoriented) as face 1.

NEGY

The element face with the outward normal most nearly parallel to the element coordinate system's negative y-axis is designated (reoriented) as face 1. .

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

POSY

The element face with the outward normal most nearly parallel to the element coordinate system's positive y-axis is designated (reoriented) as face 1.

NEGZ

(Default) The element face with the outward normal most nearly parallel to the element coordinate system's negative z-axis is designated (reoriented) as face 1.

POSZ

The element face with the outward normal most nearly parallel to the element coordinate system's positive z-axis is designated (reoriented) as face 1.

TOLER

The maximum angle (in degrees) between the outward normal face and the target axis. Default is 90.0. Lower *TOLER* values will reduce the number of faces that are considered as the basis of element reorientation.

Notes

EORIENT (p. 646) renumbers the element faces, designating the face most parallel to the XY plane of the element coordinate system (set with **ESYS (p. 685)**) as face 1 (nodes I-J-K-L, parallel to the layers in layered elements). It calculates the outward normal of each face and changes the node designation of the elements so the face with a normal most nearly parallel with and in the same general direction as the target axis becomes face 1.

The target axis, defined by *Dir*, is either the negative or positive indicated axis or the outward normal of face 1 of that element.

All layered solid elements in the selected set are considered for reorientation. The elements affected are layered structural solids (SOLID185, SOLID186), solid shell elements (SOLSH190), and layered thermal elements (SOLID278, SOLID279).

After reorienting elements, you should always display and graphically review results using the **/ESHAPE (p. 665)** command. When plotting models with many or symmetric layers, it may be useful to temporarily reduce the number of layers to two, with one layer being much thicker than the other.

You cannot use **EORIENT (p. 646)** to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Prisms and tetrahedrals are also supported for non-layered solid elements. Prisms are supported for layered solid elements, including layered SOLID185, layered SOLID186, layered SOLID278, layered SOLID279, and SOLSH190. However, layers parallel to the four-node face of the prism are not supported.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Elements>Orient Normal

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EPLOT

Produces an element display.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Produces an element display of the selected elements. In full graphics, only the element faces that have all of their corresponding nodes selected are plotted. In PowerGraphics, all element faces of the selected element set are plotted irrespective of the nodes selected. However, for both full graphics and PowerGraphics, adjacent or otherwise duplicated faces of 3D solid elements will not be displayed in an attempt to eliminate plotting of interior facets. See the **DSYS (p. 582)** command for display coordinate system issues.

This command will display curvature in midside node elements when PowerGraphics is activated (**/GRAPHICS (p. 866)**,POWER) and **/EFACET (p. 602)**,2 or **/EFACET (p. 602)**,4 are enabled. (To display curvature, two facets per edge is recommended (**/EFACET (p. 602)**,2)). When you specify **/EFACET (p. 602)**,1, PowerGraphics does not display midside nodes. **/EFACET (p. 602)** has no effect on **EPLOT (p. 648)** for non-midside node elements.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking

Utility Menu>Plot>Elements

EQSLV, *Lab*, *TOLER*, *MULT*, *--*, *KeepFile*

Specifies the type of equation solver.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Equation solver type:

SPARSE

Sparse direct equation solver. Applicable to real-value or complex-value symmetric and unsymmetric matrices. Available only for STATIC, HARMIC (full method only), TRANS (full method only), SUBSTR, and PSD spectrum analysis types (**ANTYPE (p. 162)**). Can be used for nonlinear and linear analyses, especially nonlinear analysis where indefinite matrices are frequently encountered. Well suited for contact analysis where contact status alters the mesh topology. Other typical well-suited applications are: (a) models consisting of shell/beam or shell/beam and solid elements (b) models with a multi-branch structure, such as an automobile exhaust or a turbine fan. This is an alternative to iterative solvers since it com-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bines both speed and robustness. Generally, it requires considerably more memory (~10x) than the PCG solver to obtain optimal performance (running totally in-core). When memory is limited, the solver works partly in-core and out-of-core, which can noticeably slow down the performance of the solver. See the **BCSOPTION (p. 218)** command for more details on the various modes of operation for this solver.

This solver can be run using shared-memory parallel (SMP), distributed-memory parallel (DMP), or hybrid parallel processing. For DMP, this solver preserves all of the merits of the classic or shared memory sparse solver. The total sum of memory (summed for all processes) is usually higher than the shared memory sparse solver. System configuration also affects the performance of the distributed memory parallel solver. If enough physical memory is available, running this solver in the in-core memory mode achieves optimal performance. The ideal configuration when using the out-of-core memory mode is to use one processor per machine on multiple machines (a cluster), spreading the I/O across the hard drives of each machine, assuming that you are using a high-speed network such as Infiniband to efficiently support all communication across the multiple machines.

This solver supports use of the GPU accelerator capability.

JCG

Jacobi Conjugate Gradient iterative equation solver. Available only for STATIC, HARMIC (full method only), and TRANS (full method only) analysis types (**ANTYPE (p. 162)**). Can be used for structural, thermal, and multiphysics applications. Applicable for symmetric, unsymmetric, complex, definite, and indefinite matrices. Recommended for 3D harmonic analyses in structural and multiphysics applications. Efficient for heat transfer, electromagnetics, piezoelectrics, and acoustic field problems.

This solver can be run using shared-memory parallel (SMP), distributed-memory parallel (DMP), or hybrid parallel processing.For DMP, in addition to the limitations listed above, this solver only runs in a distributed parallel fashion for STATIC and TRANS (full method) analyses in which the stiffness is symmetric and only when not using the fast thermal option (**THOPT (p. 1997)**). Otherwise, this solver disables distributed-memory parallelism at the onset of the solution, and shared-memory parallelism is used instead.

This solver supports use of the GPU accelerator capability. When using the GPU accelerator capability, in addition to the limitations listed above, this solver is available only for STATIC and TRANS (full method) analyses where the stiffness is symmetric and does not support the fast thermal option (**THOPT (p. 1997)**).

ICCG

Incomplete Cholesky Conjugate Gradient iterative equation solver. Available for STATIC, HARMIC (full method only), and TRANS (full method only) analysis types (**ANTYPE (p. 162)**). Can be used for structural, thermal, and multiphysics applications, and for symmetric, unsymmetric, complex, definite, and indefinite matrices. The ICCG solver requires more memory than the JCG solver, but is more robust than the JCG solver for ill-conditioned matrices.

This solver can only be run in shared-memory parallel mode. If it is run in DMP mode, this solver disables distributed-memory parallelism at the onset of the solution, and shared-memory parallelism is used instead

This solver does not support use of the GPU accelerator capability.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

QMR

Quasi-Minimal Residual iterative equation solver. Available for the HARMIC (full method only) analysis type (**ANTYPE (p. 162)**). Can be used for symmetric, complex, definite, and indefinite matrices. The QMR solver is more stable than the ICCG solver.

This solver can only be run in shared memory parallel mode and only supports 1 core. If it is run in DMP mode, this solver disables distributed-memory parallelism at the onset of the solution, and shared-memory parallelism is used instead

This solver does not support use of the GPU accelerator capability.

PCG

Preconditioned Conjugate Gradient iterative equation solver (licensed from Computational Applications and Systems Integration, Inc.). Requires less disk file space than SPARSE and is faster for large models. Useful for plates, shells, 3D models, large 2D models, and other problems having symmetric or unsymmetric, sparse matrices. Such matrices are typically positive definite or indefinite for symmetric full harmonic analyses that solve a symmetric complex number matrix system. The PCG solver can be used for:

- single-field structural analyses involving symmetric or unsymmetric matrices
- single-field acoustic analyses with pressure as the degree of freedom
- single-field thermal analyses involving symmetric or unsymmetric matrices

Requires twice as much memory as JCG. Available only for analysis types (**ANTYPE (p. 162)**): STATIC, TRANS (full method only), HARMIC (full method only), or MODAL (with PCG Lanczos option only). Also available for the use pass of substructure analyses (MATRIX50). The PCG solver can robustly handle models that involve the use of constraint and/or coupling equations (**CE (p. 297)**, **CEINTF (p. 305)**, **CP (p. 421)**, **CPINTF (p. 425)**, and **CERIG (p. 310)**). With this solver, you can use the **MSAVE (p. 1183)** command to obtain a considerable memory savings.

The PCG solver can handle ill-conditioned problems by using a higher level of difficulty (see **PCGOPT (p. 1365)**). Ill-conditioning arises from elements with high aspect ratios, contact, and plasticity.

This solver can be run in shared-memory parallel or distributed-memory parallel mode. In DMP mode, this solver preserves all of the merits of the classic or shared-memory PCG solver. The total sum of memory (summed for all processes) is about 30% more than the shared-memory PCG solver.

This solver supports use of the GPU accelerator capability.

TOLER

Iterative solver tolerance value. Used only with the Jacobi Conjugate Gradient, Incomplete Cholesky Conjugate Gradient, Pre-conditioned Conjugate Gradient, and Quasi-Minimal Residual equation solvers. For the (symmetric or unsymmetric) PCG solver, the default is 1.0E-8. When using the PCG Lanczos mode extraction method, the default solver tolerance value is 1.0E-4. For the JCG and ICCG solvers with symmetric matrices, the default is 1.0E-8. For the PCG solver used in a full harmonic analysis, the default tolerance is 1.0E-6. For the JCG and ICCG solvers with unsymmetric matrices,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and for the QMR solver, the default is 1.0E-6. Iterations continue until the SRSS norm of the residual is less than *TOLER* times the norm of the applied load vector. For the PCG solver in the linear static analysis case, 3 error norms are used. If one of the error norms is smaller than *TOLER*, and the SRSS norm of the residual is smaller than 1.0E-2, convergence is assumed to have been reached. See Iterative Solver in the *Mechanical APDL Theory Reference* for details.

Note:

When used with the Pre-conditioned Conjugate Gradient equation solver, *TOLER* can be modified between load steps (this is typically useful for nonlinear analysis).

If a *Lev_Diff* value of 5 is specified on the **PCGOPT (p. 1365)** command (either program- or user-specified), *TOLER* has no effect on the accuracy of the obtained solution from the PCG solver; a direct solver is used when *Lev_Diff* = 5.

MULT

Multiplier (defaults to 2.5 for nonlinear analyses; 1.0 for linear analyses) used to control the maximum number of iterations performed during convergence calculations. Used only with the Pre-conditioned Conjugate Gradient equation solver (PCG). The maximum number of iterations is equal to the multiplier (*MULT*) times the number of degrees of freedom (DOF). If *MULT* is input as a negative value, then the maximum number of iterations is equal to abs(*MULT*). Iterations continue until either the maximum number of iterations or solution convergence has been reached. In general, the default value for *MULT* is adequate for reaching convergence. However, for ill-conditioned matrices (that is, models containing elements with high aspect ratios or material type discontinuities) the multiplier may be used to increase the maximum number of iterations used to achieve convergence. The recommended range for the multiplier is 1.0 ≤ *MULT* ≤3.0. Normally, a value greater than 3.0 adds no further benefit toward convergence, and merely increases time requirements. If the solution does not converge with 1.0 ≤ *MULT* ≤3.0, or in less than 10,000 iterations, then convergence is highly unlikely and further examination of the model is recommended. Rather than increasing the default value of *MULT*, consider increasing the level of difficulty (*Lev_Diff*) on the **PCGOPT (p. 1365)** command.

--

Unused field.

KeepFile

Determines whether files from a SPARSE solver run should be deleted or retained. Applies only to *Lab* = SPARSE for static and full transient analyses.

DELE

Deletes all files from the SPARSE solver run, including the factorized file, `.DSPsymb`, upon **FINISH (p. 758)** or **/EXIT (p. 708)** (default).

KEEP

Retains all necessary files from the SPARSE solver run, including the `.DSPsymb` file, in the working directory.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

The sparse direct solver is the default solver for all analyses, with the exception of modal/buckling analyses.

For modal/buckling analyses, there is no default solver. You must specify a solver with the **MODOPT (p. 1140)** or **BUCOPT (p. 280)** command. The specified solver automatically chooses the required internal equation solver (for example, **MODOPT (p. 1140)**,LANPCG automatically uses **EQSLV (p. 648)**,PCG internally, and **BUCOPT (p. 280)**,LANB automatically uses **EQSLV (p. 648)**,SPARSE internally).

Notes

The selection of a solver can affect the speed and accuracy of a solution. For a more detailed discussion of the merits of each solver, see Solution in the *Basic Analysis Guide* .

This command is also valid in PREP7.

Distributed-Memory Parallel (DMP) Restriction

All equation solvers are supported in a DMP analysis. However, the SPARSE and PCG solvers are the only distributed solvers that always run a fully distributed solution. The JCG solver runs in a fully distributed mode in some cases; in other cases, it does not. The ICCG and QMR solvers are not distributed solvers; therefore, you will not see the full performance improvements with these solvers that you would with a fully distributed solution.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Sol'n Options

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Sol'n Controls>Sol'n Options

---

ERASE

Explicitly erases the current display.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Similar to a hardware screen erase key. Useful during an "immediate" display to erase the screen without a replot so that the display continues on a clean screen. This action is automatically included in commands such as **NPLOT (p. 1254)** and **EPLOT (p. 648)**.

If the **/NOERASE (p. 1249)** command is active, issuing the erase command will simply clear the display area. Subsequent replots will provide the display previously generated by the **/NOERASE (p. 1249)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase Screen

/ERASE

Specifies that the screen is to be erased before each display.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Command Default

Previous **/ERASE (p. 653)** or **/NOERASE (p. 1249)** setting will be used. Initially defaults to **/ERASE (p. 653)** setting.

Notes

Erase occurs with the next display request, but before the display is actually started. **/NOERASE (p. 1249)** can be used to suppress the automatic screen erase.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase Between Plots

EREAD,*Fname*,*Ext*,--

Reads elements from a file.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to ELEM if *Fname* is blank.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- -

Unused field.

Notes

This read operation is not necessary in a standard anlaysis run but is provided as a convenience for those wanting to read a coded element file, such as from another mesh generator or from a CAD/CAM program.

Data should be formatted as generated via **EWRITE (p. 703)**.

If issuing **EREAD (p. 653)** to acquire element information generated from **EWRITE (p. 703)**, you must also issue **NREAD (p. 1256)** before the **EREAD (p. 653)** command. The element types (**ET (p. 686)**) must be defined before the file is read so that the file may be read properly. Only elements that are specified via **ERRANG (p. 660)** are read from the file. Also, only elements that are fully attached to the nodes specified via **NRRANG (p. 1265)** are read from the file.

Elements are assigned numbers consecutively as read from the file, beginning with the current highest database element number plus one. The file is rewound before and after reading. Reading continues until the end of the file.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Read Elem File

---

EREFINE, *NE1*, *NE2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified elements.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NE1*, *NE2*, *NINC

Elements (*NE1* to *NE2* in increments of *NINC*) around which the mesh is to be refined. *NE2* defaults to *NE1*, and *NINC* defaults to 1. If *NE1* = ALL, *NE2* and *NINC* are ignored and all selected elements are used for refinement. If *NE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NE1* (*NE2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH

Depth of mesh refinement in terms of number of elements outward from the indicated elements, *NE1* to *NE2* (defaults to 0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

POST

Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF

No postprocessing will be done.

SMOOTH

Smoothing will be done. Node locations may change.

CLEAN

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (The program ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

OFF

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

EREFINE (p. 654) refines all area elements and tetrahedral volume elements adjacent to the specified elements.

Mesh refinement is not available on a solid model containing initial conditions at nodes (**IC (p. 921)**), coupled nodes (**CP (p. 421)** family of commands), constraint equations (**CE (p. 297)** family of commands), or boundary conditions or loads applied directly to any of its nodes or elements. This restriction applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. If you have detached the mesh from the solid model, disable postprocessing cleanup or smoothing (*POST* = OFF) after refinement to preserve the element attributes.

For more information about mesh refinement, see Revising Your Model.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>All

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EREINF,*KOffALim*,*KTri*

Generates reinforcing elements from selected existing (base) elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KOffALim

Enable or disable the limit of the angle between a MESH200 element and a base element. Valid for the mesh-independent method only.

0 – Enable the angle limit (default).

1 – Disable the angle limit.

KTri

Specify the shape of 3D smeared reinforcing members. Valid for the mesh-independent method only.

0 – Generate quad-dominant (mixed quadrilateral and triangular) reinforcing members (default).

1 – Generate triangular reinforcing members only.

Notes

EREINF (p. 656) generates reinforcing elements (REINF263, REINF264 and REINF265) directly from selected base elements (that is, existing standard structural elements in your model). The command scans all selected base elements and generates (if necessary) a compatible reinforcing element type for each base element. (You can select a combination of different base element types.)

Before issuing **EREINF (p. 656)**, first define the reinforcing geometry, material, and orientation via one of two methods:

Mesh-Independent Method:

Use MESH200 elements to temporarily represent the geometry of the reinforcing fibers or smeared reinforcing surfaces. Define additional data including material, fiber cross-section area, fiber spacing, and fiber orientation via reinforcing sections with the mesh pattern (**SECDATA (p. 1655)**) and assign the sections to corresponding MESH200 elements. (Predefining the reinforcing element type (**ET (p. 686)**) is not required.)

Standard Method:

Define reinforcing section types (**SECTYPE (p. 1697)**) with standard reinforcing location patterns (**SECDATA (p. 1655)**). The standard reinforcing location input are given with respect to the selected base elements; therefore, a change in the base mesh may require redefining the (mesh-dependent) reinforcing section types.

Standard element-definition commands (such as **ET (p. 686)** and **E (p. 591)**) are not used for defining reinforcing elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EREINF (p. 656) creates no new nodes. The reinforcing elements and the base elements share the common nodes.

Elements generated by **EREINF (p. 656)** are not associated with the solid model.

After **EREINF (p. 656)** executes, you can issue **ETLIST (p. 702)**, **ELIST (p. 619)**, or **EPLOT (p. 648)** to verify the newly created reinforcing element types and elements.

Reinforcing elements do not account for any subsequent modifications made to the base elements. Ansys, Inc. recommends issuing **EREINF (p. 656)** only *after* the base elements are finalized. If you delete or modify base elements (via **EDELE (p. 593)**, **EMODIF (p. 626)**, **ETCHG (p. 699)**, **EMID (p. 624)**, **EORIENT (p. 646)**, **NUMMRG (p. 1295)**, or **NUMCMP (p. 1291)**, for example), remove all affected reinforcing elements and reissue **EREINF (p. 656)** to avoid inconsistencies.

If you define reinforcing via the mesh-independent method, **EREINF (p. 656)** creates new reinforcing sections containing details of the created reinforcing elements, then applies them to all newly generated reinforcing elements. The number of new reinforcing sections depends on the number of new reinforcing elements. (You can examine the properties of new sections (**SLIST (p. 1795)**).) The program sets the ID number of the newest reinforcing section to the highest section ID number in the model. After issuing **EREINF (p. 656)**, the command shows the highest-numbered IDs (element type, element, and section). Do not overwrite a new reinforcing section when defining subsequent sections.

For the 3D smeared-reinforcing element (REINF265) with the mesh-independent method, you can select the shape of the reinforcing members via *KTri*. The default behavior (*KTri* = 0) generates quad-dominant members (primarily quadrilaterals but with some triangles).

EREINF (p. 656) can generate the reinforcing elements with thermal properties if the base elements are thermal solid elements (SOLID278 or SOLID279):

- If using the mesh-independent method for defining reinforcing, apply element body-force loading (**BFE (p. 237)**,,HGEN) or nodal body-force loading (**BF (p. 221)**,,HGEN) on the MESH200 elements.
- If using the standard method for defining reinforcing, apply element body-force loading (**BFE (p. 237)**,,HGEN) on the reinforcing members directly. (Do not apply *nodal* body-force loading (**BF (p. 221)**,,HGEN).)

If performing a subsequent structural analysis after the thermal analysis, **EREINF (p. 656)** can convert the reinforcing elements for the structural analysis:

1. Convert the thermal base elements to the appropriate structural element (**ET (p. 686)** or **EMODIF (p. 626)**).
2. Select the reinforcing elements only.
3. Issue **EREINF (p. 656)**.

   Result: The selected reinforcing elements are converted to elements compatible with the converted base elements.

Solution accuracy can be affected if the volume ratio between reinforcing elements and base elements is high, particularly when body loading (such as heat generation) is applied via the reinforcing elements. If the program detects a high volume ratio of reinforcing elements generated via the mesh-independent method, it issues a warning message and saves affected base and MESH200 elements (used to generate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the reinforcing elements) into a component for close model inspection. Only the volume of reinforcing elements created by the *current* **EREINF (p. 656)** command is considered in the volume-ratio calculation (that is, no volume accumulation occurs over multiple **EREINF (p. 656)** commands).

For more information, see Reinforcing and Direct Element Embedding in the *Structural Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

ERESX, *Key*

Specifies extrapolation of integration-point results.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Extrapolation key:

DEFA

If element is fully elastic (no active plasticity, creep, or swelling nonlinearities), extrapolate the integration-point results to the nodes. If any portion of the element is plastic (or other active material nonlinearity), copy the integration-point results to the nodes (default).

YES

Extrapolate the linear portion of the integration-point results to the nodes and copy the nonlinear portion (for example, plastic strains).

NO

Copy the integration-point results to the nodes.

Command Default

Extrapolate integration-point results to the nodes for all elements except those with active plasticity, creep, or swelling nonlinearities (default).

For coupled pore-pressure-thermal elements (CPT212, CPT213, CPT215, CPT216, CPT217), the default behavior is to copy integration-point results to the nodes.

Notes

Specifies whether the solution results at the element-integration points are extrapolated or copied to the nodes for element and nodal postprocessing. Structural stresses, elastic and thermal strains, field gradients, and fluxes are affected. Nonlinear data (such as plastic, creep, and swelling strains) are always copied to the nodes, never extrapolated. For shell elements, **ERESX (p. 658)** applies only to integration-point results in the in-plane directions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Extrapolation occurs in the element-solution coordinate system. For elements allowing different solution systems at integration points (such as SHELL281 and SOLID186), extrapolation can produce unreliable results when the solution coordinate systems in each element differ significantly. (Varying element-solution coordinate systems can be created via the **ESYS (p. 685)** command or from large deformation.) Examine results carefully in such cases, and disable extrapolation if necessary.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Integration Pt

Main Menu>Solution>Load Step Opts>Output Ctrls>Integration Pt

---

ERNORM, *Key*

Controls error estimation calculations.

POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Control key:

ON

Perform error estimation (default). This option is not valid for PowerGraphics.

OFF

Do not perform error estimation.

Command Default

Error estimation calculations are performed by default unless PowerGraphics is enabled (**/GRAPHICS (p. 866)**,POWER).

Notes

Especially for thermal analyses, program speed increases if error estimation is suppressed. Therefore, it might be desirable to use error estimation only when needed. The value of the **ERNORM (p. 659)** key is not saved on `file.db`. Consequently, you need to reissue the **ERNORM (p. 659)** key after a **RESUME (p. 1601)** if you wish to deactivate error estimation again.

Menu Paths

Main Menu>General Postproc>Options for Outp

Utility Menu>List>Results>Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ERRANG, *EMIN*, *EMAX*, *EINC*

Specifies the element range to be read from a file.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

EMIN*, *EMAX*, *EINC

Elements with numbers from *EMIN* (defaults to 1) to *EMAX* (defaults to 999999999) in steps of *EINC* (defaults to 1) will be read.

Notes

Defines the element number range to be read (**EREAD (p. 653)**) from the element file. If a range is also implied from the **NRRANG (p. 1265)** command, only those elements satisfying both ranges will be read.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Read Elem File

ESCHECK, *Sele*, *Levl*, *Defkey*

Perform element shape checking for a selected element set.

PREP7 (p. 22): Database (p. 22)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sele

Specifies whether to select elements for checking:

(blank)

List all warnings/errors from element shape checking.

ESEL

Select the elements based on the *.Levl* criteria specified below.

Levl

WARN

Select elements producing warning and error messages.

ERR

Select only elements producing error messages (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Defkey

Specifies whether check should be performed on deformed element shapes.

0

Do not update node coordinates before performing shape checks (default).

1

Update node coordinates using the current set of deformations in the database.

Notes

Shape checking will occur according to the current **SHPP (p. 1787)** settings. Although **ESCHECK (p. 660)** is valid in all processors, *Defkey* uses the current results in the database. If no results are available a warning will be issued.

This command is also valid in PREP7, SOLUTION and POST1.

Menu Paths

Main Menu>General Postproc>Check Elem Shape>Sel Warning/Error Elements

---

ESEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Selects a subset of elements.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item

Label identifying data, see Table 110: ESEL - Valid Item and Component Labels (p. 663). Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to ELEM. If *Item* = STRA (straightened), elements are selected whose midside nodes do not conform to the curved line or non-flat area on which they should lie. (Such elements are sometimes formed during volume meshing (**VMESH (p. 2140)**) in an attempt to avoid excessive element distortion.) You should graphically examine any such elements to evaluate their possible effect on solution accuracy.

Comp

Component of the item (if required). Valid component labels are shown in Table 110: ESEL - Valid Item and Component Labels (p. 663) below.

VMIN

Minimum value of item range. Ranges are element numbers, attribute numbers, load values, or result values as appropriate for the item. A component name (as specified via the **CM (p. 356)** command) can also be substituted for *VMIN* (in which case *VMAX* and *VINC* are ignored).

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN* for input values.

For result values, *VMAX* defaults to infinity if *VMIN* is positive, or to zero if *VMIN* is negative.

VINC

Value increment within range. Used only with integer ranges (such as for element and attribute numbers). Defaults to 1. *VINC* cannot be negative.

KABS

Absolute value key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Check sign of value during selection.

1

Use absolute value during selection (sign ignored).

Command Default

All elements are selected.

Notes

Selects elements based on values of a labeled item and component. For example, to select a new set of elements based on element numbers 1 through 7, use **ESEL (p. 661)**,S,ELEM,,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **ELIST (p. 619)**,ALL. Only data identified by element number are selected. Selected data are internally flagged; no actual removal of data from the database occurs. Different element subsets cannot be used for different load steps (**SOLVE (p. 1822)**) in a **/SOLU (p. 1821)** sequence. The subset used in the first load step is used for all subsequent load steps regardless of subsequent **ESEL (p. 661)** specifications.

This command is valid in any processor.

Elements crossing the named path (**PATH (p. 1354)**) are selected. This option is available only in PREP7 and POST1. If no geometry data has been mapped to the path (via **PMAP (p. 1449)** and **PDEF (p. 1372)**, for example), the path assumes the default mapping option (**PMAP (p. 1449)**,UNIFORM) to map the geometry prior to selecting the elements. If an invalid path name is given, the **ESEL (p. 661)** command is ignored (and the status of selected elements is unchanged). If no elements are crossing the path, the **ESEL (p. 661)** command returns zero elements selected.

For selections based on non-integer numbers (coordinates, results, etc.), items that are within the range *VMIN* -*Toler* and *VMAX* + *Toler* are selected. The default tolerance *Toler* is based on the relative values of *VMIN* and *VMAX* as follows:

- If *VMIN* = *VMAX*, *Toler* = 0.005 x *VMIN*.
- If *VMIN* = *VMAX* = 0.0, *Toler* = 1.0E-6.
- If *VMAX* ≠ *VMIN*, *Toler* = 1.0E-8 x (*VMAX* - *VMIN*).

To override this default and specify *Toler* explicitly, issue the **SELTOL (p. 1710)** (p. 1710) command.

Table 110: ESEL (p. 661) - Valid Item and Component Labels

Valid Item and Component Labels ESEL (p. 661), *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Item	Comp	Description
ELEM		Element number.
ADJ		Elements adjacent to element *VMIN* (*VMAX* and *VINC* fields are ignored). Only elements (of the same dimensionality) adjacent to

Valid Item and Component Labels ESEL (p. 661), *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Item	Comp	Description
		lateral faces are considered. Progression continues until edge of model or until more than two elements are adjacent at a face.
CENT	X, Y, Z	X, Y, or Z location in the active coordinate system.
TYPE		Element type number.
ENAME		Element name (or identifying number).
MAT		Material ID number.
REAL	(blank)	Real constant number.
	GCN	General contact/target elements (select all elements identified by real constant set number = 0). Remaining fields (*VMIN*, *VMAX*, etc.) are ignored.
	BASE	All split contact pairs (**CNCHECK (p. 382)**,SPLIT/DMP) that are associated with the current real constant set number.
ESYS		Element coordinate system number.
OVER		Overlapping contact elements created during contact pair splitting (**CNCHECK (p. 382)**,SPLIT/DMP)
LIVE		Active elements (**EALIVE (p. 592)**). *VMIN* and *VMAX* fields are ignored.
LAYER		Layer number (where only composite elements with a nonzero thickness for the requested layer number are included) (**LAYER (p. 1015)**).
SEC	(blank)	Cross section ID number (**SECNUM (p. 1685)**)
	MAT	Selects the elements with the requested MAT ID specified via *VMIN* and *VMAX* as attached to the section.
STRA		Straightened. See the description of the *Item* argument above.
SFE	PRES	Element pressure.
	CONV	Element convection bulk temperature.
	HFLUX	Element heat flux.
	FSI	Element (acoustic) fluid-structure interaction flag.
	IMPD	Element (acoustic) impedance.
	SHLD	Surface normal velocity or acceleration (acoustic analysis).
	MXWF	Element Maxwell force flag.
	CHRGS	Electric surface charge density.
	INF	Element infinite surface flag.
	DFLUX	Element diffusion flux.
BFE	TEMP	Element temperature.
	FLUE	Element fluence.
	HGEN	Element heat generation rate.
	JS	Element current density, magnitude only.
	MVDI	Element magnetic virtual displacements flag.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Item and Component Labels ESEL (p. 661), *Type, Item, Comp, VMIN, VMAX, VINC, KABS*

Item	Comp	Description
	DGEN	Element diffusing substance generation rate.
	CHRGD	Electric charge density.
PATH	*Lab*	Selects all elements being crossed by the path with name *Lab* (**PATH (p. 1354)**). If *Lab* = ALL, all elements related to all defined paths are selected.
	Valid item and component labels for element result values are:	
ETAB	*Lab*	Any user-defined element table label (**ETABLE (p. 687)**).

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts

Utility Menu>Select>Entities

/ESHAPE, *SCALE*, *KEY*

Displays elements with shapes determined from the real constants, section definition, or other inputs.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SCALE

Scaling factor:

0

Use simple display of line and area elements. This value is the default.

1

Use real constants, section definition, or other information to form a solid shape display of the applicable elements.

FAC

Multiply certain real constants, such as thickness, by *FAC* (where *FAC* > 0.01) and use them to form a solid shape display of elements.

KEY

Current shell thickness key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Use current thickness in the displaced solid shape display of shell elements (valid for SHELL181, SHELL208, SHELL209, and SHELL281). This value is the default.

1

Use initial thickness in the displaced solid shape display of shell elements.

Command Default

Use simple display of line and area elements (*SCALE* = 0).

Notes

The **/ESHAPE (p. 665)** command allows beams, shells, current sources, and certain special-purpose elements or elements with special options to be displayed as solids with the shape determined from the real constants, section types, or other information. Elements are displayed via the **EPLOT (p. 648)** command. No checks for valid or complete input are made for the display.

Following are details about using this command with various element types:

- COMBIN14, COMBIN39, and MASS21 are displayed with a graphics icon, with the offset determined by the real constants and KEYOPT settings.
- BEAM188, BEAM189, PIPE288, PIPE289 and ELBOW290 are displayed as solids with the shape determined via the section-definition commands (**SECTYPE (p. 1697)** and **SECDATA (p. 1655)**). The arbitrary section option (*Subtype* = ASEC) has no definite shape and appears as a thin rectangle to indicate the orientation; the thin side represents the beam Y axis and the thick (longer) side represents the Z axis. The length of thick side is determined by *TKZ*. If *TKZ* = 0, the area is used to determine the length of thick side. The thin side is scaled by the fraction of the thick side, regardless of *TKY*. If the offsets are defined via *CGY*/*CGZ* (**SECDATA (p. 1655)** or **SECOFFSET (p. 1685)**,USER,*OFFSETY*,*OFFSETZ*), they are applied to the plot without scaling. The elements are displayed with internal lines representing the cross-section mesh.
- Reduced-integration and lower-order shells (SHELL181 and SHELL208 with KEYOPT(3)=0) are displayed with uniform thickness, evaluated at the centroid, to reflect the element behavior.
- SOLID272 and SOLID273 are displayed as solids with the shape determined via the section-definition commands (**SECTYPE (p. 1697)** and **SECDATA (p. 1655)**). The 2D master plane is revolved around the prescribed axis of symmetry.
- PLANE182, PLANE183, PLANE222, and PLANE223 with KEYOPT(3) = 6 are displayed as 3D solids with the shape determined by nodal locations and displacements at the nodes.
- Contour plots are available for these elements in postprocessing for PowerGraphics only (**/GRAPHICS (p. 866)**,POWER). To view 3D deformed shapes for the elements, issue **OUTRES (p. 1336)**,MISC or **OUTRES (p. 1336)**,ALL for static or transient analyses. To view 3D mode shapes for a modal or eigenvalue buckling analysis, expand the modes with element results calculation ON (*Elcalc* = YES for **MXPAND (p. 1203)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- SOURC36, CIRCU124, and TRANS126 elements always plot using **/ESHAPE (p. 665)** when PowerGraphics is activated (**/GRAPHICS (p. 866)**,POWER).

In most cases, **/ESHAPE (p. 665)** renders a thickness representation of your shell, plane and layered elements more readily in PowerGraphics (**/GRAPHICS (p. 866)**,POWER). This type of representation employs PowerGraphics to generate the enhanced representation, and will often provide no enhancement in Full Graphics (**/GRAPHICS (p. 866)**,FULL). This is especially true for POST1 results displays, where **/ESHAPE (p. 665)** is not supported for most element types with FULL graphics.

When PowerGraphics is active, **/ESHAPE (p. 665)** may degrade the image if adjacent elements have overlapping material, such as shell elements which are not co-planar. Additionally, if adjacent elements have different thicknesses, the polygons depicting the connectivity between the "thicker" and "thinner" elements along the shared element edges may not always be displayed.

For POST1 results displays (such as **PLNSOL (p. 1425)**), the following limitations apply:

- If you issue **RSYS (p. 1639)**,SOLU before reviewing results for beam or pipe elements, contour plots for displacement (for example, **PLNSOL (p. 1425)**,U,*X* and **PLDISP (p. 1400)**) do not appear in the solution coordinate system when **/ESHAPE (p. 665)** is active. Instead, the contours appear in the global Cartesian coordinate system.

- When shell elements are not co-planar, the resulting **PLNSOL (p. 1425)** display with **/ESHAPE (p. 665)** will actually be a **PLESOL (p. 1401)** display as the non-coincident pseudo-nodes are not averaged. Additionally, **/ESHAPE (p. 665)** should not be used with coincident elements because the plot may incorrectly average the displacements of the coincident elements.

- When nodes are initially coincident and PowerGraphics is active, duplicate polygons are eliminated to conserve display time and disk space. The command may degrade the image if initially coincident nodes have different displacements. The tolerance for determining coincidence is 1E-9 times the model's bounding box diagonal.

- If you want to view solution results (**PLNSOL (p. 1425)**, etc.) on layered elements (such as SHELL181, SOLSH190, SOLID185 Layered Solid, SOLID186 Layered Solid, SHELL208, SHELL209, SHELL281, and ELBOW290), set KEYOPT(8) = 1 for the layer elements so that the data for all layers is stored in the results file.

- You can plot the through-thickness temperatures of elements SHELL131 and SHELL132 regardless of the thermal DOFs in use by issuing the **PLNSOL (p. 1425)**,TEMP command (with PowerGraphics and **/ESHAPE (p. 665)** active).

- The **/ESHAPE (p. 665)**,1 and **/ESHAPE (p. 665)**,*FAC* commands are incompatible with the **/CYCEXPAND (p. 454)** command used in cyclic symmetry analyses.

- The **/ESHAPE (p. 665)**,1 command does not support velocity and acceleration results for elements PLANE182, PLANE183, PLANE222, or PLANE223 with KEYOPT(3) = 6.

- For coupled-field elements PLANE222 and PLANE223 with KEYOPT(3) = 6, the **/ESHAPE (p. 665)** command can be used to visualize structural results. Non-structural results visualization is limited to gradient and flux displays supported with PowerGraphics.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

ESIZE, *SIZE*, *NDIV*

Specifies the default number of line divisions.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIZE

Default element edge length on surface boundaries (that is, lines). Divisions are automatically calculated (rounded upward to next integer) from line lengths. If *SIZE* is zero (or blank), use *NDIV*.

NDIV

Default number of element divisions along region boundary lines. Not used if *SIZE* is input.

Notes

Specifies the default number of line divisions (elements) to be generated along the region boundary lines. The number of divisions may be defined directly or automatically calculated. Divisions defined directly for any line (**LESIZE (p. 1041)**, **KESIZE (p. 978)**, etc.) are retained. For adjacent regions, the divisions assigned to the common line for one region are also used for the adjacent region. See the **MOPT (p. 1151)** command for additional meshing options.

For free meshing operations, if smart element sizing is being used (**SMRTSIZE (p. 1811)**) and **ESIZE (p. 668)**,*SIZE* has been specified, *SIZE* will be used as a starting element size, but will be overridden (that is, a smaller size may be used) to accommodate curvature and small features.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Size

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts

---

ESLA, *Type*

Selects those elements associated with the selected areas.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of element select:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Notes

Selects area elements belonging to meshed (**AMESH (p. 121)**), selected (**ASEL (p. 192)**) areas.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ESLL, *Type*

Selects those elements associated with the selected lines.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of element select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Selects line elements belonging to meshed (**LMESH (p. 1057)**), selected (**LSEL (p. 1074)**) lines.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ESLN, *Type*, *EKEY*, *NodeType*

Selects those elements attached to the selected nodes.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of element selected:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

EKEY

Node set key:

0

Select element if *any* of its nodes are in the selected nodal set (default).

1

Select element only if *all* of its nodes are in the selected nodal set.

NodeType

Label identifying type of nodes to consider when selecting:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Select elements considering all of their nodes (default).

ACTIVE

Select elements considering only their active nodes. An active node is a node that contributes DOFs to the model.

INACTIVE

Select elements considering only their inactive nodes (such as orientation or radiation nodes).

CORNER

Select elements considering only their corner nodes.

MID

Select elements considering only their midside nodes.

Notes

ESLN (p. 670) selects elements which have any (or all *EKEY*) *NodeType* nodes in the currently-selected set of nodes. Only elements having nodes in the currently-selected set can be selected.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts

Utility Menu>Select>Entities

ESLV, *Type*

Selects elements associated with the selected volumes.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of element selected:

S

Select a new set (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Notes

Selects volume elements belonging to meshed (**VMESH (p. 2140)**), selected (**VSEL (p. 2163)**) volumes.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

ESOL, *NVAR*, *ELEM*, *NODE*, *Item*, *Comp*, *Name*

Specifies element data to be stored from the results file.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number assigned to this variable (2 to *NV* (**NUMVAR (p. 1302)**)). Overwrites any existing results for this variable.

ELEM

Element for which data are to be stored. If *ELEM* = P, graphical picking is enabled (valid only in the GUI).

NODE

Node number on this element for which data are to be stored. If blank, store the average element value (except for *FMAG* values, which are summed instead of averaged). If *NODE* = P, graphical picking is enabled (valid only in the GUI).

Item

Label identifying the item. General item labels are shown in Table 111: ESOL - General Result Item and Component Labels (p. 673). Some items also require a component label. For selected result output, specify SRES and see Table 112: ESOL - Selected Result (SRES) Component Labels (p. 679).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Comp

Component of the item (if required). General component labels are shown in Table 111: ESOL - General Result Item and Component Labels (p. 673) below. If *Comp* is a sequence number (*n*), the *NODE* field is ignored.

Name

32-character name for identifying the item on the printout and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

See Table 111: ESOL - General Result Item and Component Labels (p. 673) for a list of item and component labels for element (excluding line element) results. See Table 112: ESOL - Selected Result (SRES) Component Labels (p. 679) for a list of valid selected result (*Item* = SRES) components.

ESOL (p. 672) defines element results data to be stored from a results file (**FILE (p. 752)**). Not all items are valid for all elements. To see the available items for a given element, refer to the input and output summary tables in the documentation for that element.

Two methods of data access are available via the **ESOL (p. 672)** command. You can access some data by using a generic label (*component name method*), while others require a label and number (*sequence number method*).

Use the component name method to access general element data (that is, element data generally available to most element types or groups of element types). Element results are in the element coordinate system, except for layered elements where results are in the layer coordinate system. Element forces and moments are in the nodal coordinate system. Results are obtainable for an element at a specified node. Further location specifications can be made for some elements via **SHELL (p. 1784)**, **LAYERP26 (p. 1016)**, and **FORCE (p. 772)**.

The sequence number method is required for data that is not averaged (such as pressures at nodes and temperatures at integration points), or data that is not easily described generically (such as all derived data for structural line elements and contact elements, all derived data for thermal line elements, and layer data for layered elements).

In a 2D to 3D analysis, this command not supported in the POST26 postprocessor and is ignored.

Table 111: ESOL (p. 672) - General Result Item and Component Labels

Component Name Method		
Item	**Comp**	**Description**
S	X, Y, Z, XY, YZ, XZ	Component stress.
	1, 2, 3	Principal stress.
	INT	Stress intensity.
	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
	1, 2, 3	Principal elastic strain.
	INT	Elastic strain intensity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Name Method		
Item	**Comp**	**Description**
	EQV	Elastic equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
	1, 2, 3	Principal thermal strain.
	INT	Thermal strain intensity.
	EQV	Thermal equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
	1, 2, 3	Principal plastic strain.
	INT	Plastic strain intensity.
	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
	1,2,3	Principal creep strain.
	INT	Creep strain intensity.
	EQV	Creep equivalent strain.
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain.
	1, 2, 3	Principal diffusion strain.
	INT	Diffusion strain intensity.
	EQV	Diffusion equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
	SRAT	Stress state ratio.
	HPRES	Hydrostatic pressure.
	EPEQ	Accumulated equivalent plastic strain.
	CREQ	Accumulated equivalent creep strain.
	PSV	Plastic state variable.
	PLWK	Plastic work/volume.
SEND	ELASTIC[a]	Elastic strain energy density. (For viscoelastic and sintering materials, the stored energy.)
	PLASTIC	Plastic strain energy density.
	CREEP	Creep strain energy density.
	DAMAGE	Damage strain energy density.
	VDAM[a]	Viscoelastic dissipation energy density.
	VREG	Visco-regularization strain energy density.
	DISS	Structural-thermal dissipation.
	ENTO	Total strain energy density (sum of ELASTIC, PLASTIC, and CREEP strain energy densities).
CDM	DMG	Damage variable.
	LM	Maximum previous strain energy for virgin material.
GKS	X	Gasket component stress (also gasket pressure).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Name Method		
Item	**Comp**	**Description**
GKD	X	Gasket component total closure.
GKDI	X	Gasket component total inelastic closure.
GKTH	X	Gasket component thermal closure.
SS	X, XY, XZ	Interface traction (stress).
SD	X,XY,XZ	Interface separation.
CONT	STAT[b]	Contact status.
	PENE	Contact penetration.
	PRES	Contact pressure.
	SFRIC	Contact friction stress.
	STOT	Contact total stress (pressure plus friction).
	SLIDE	Contact sliding distance.
	GAP	Contact gap distance.
	FLUX	Total heat flux at contact surface.
	CNOS	Total number of contact status changes during substep.
	FPRS	Fluid penetration pressure.
TG[c]	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF[c]	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum.
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum.
FMAG	X, Y, Z, SUM	Component electromagnetic forces or vector sum.
P	X, Y, Z, SUM	Poynting vector components or vector sum
F	X, Y, Z	Component structural force.
M	X, Y, Z	Component structural moment.
HEAT[c]		Heat flow.
FLOW		Fluid flow.
AMPS		Current flow.
FLUX		Magnetic flux.
CSG	X, Y, Z	Component magnetic current segment.
RATE		Diffusion flow rate.
SENE		"Stiffness" energy.
STEN		Elemental energy dissipation due to stabilization.
KENE		Kinetic energy.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Name Method		
Item	**Comp**	**Description**
ASENE		Amplitude stiffness energy.
PSENE		Peak stiffness energy.
AKENE		Amplitude kinetic energy.
PKENE		Peak kinetic energy.
DENE		Damping energy.
WEXT[d]		Work due to external load.
AENE		Artificial energy due to hourglass control/drill stiffness or due to contact stabilization.
JHEAT		Element Joule heat generation.
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).
JS	X, Y, Z	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low-frequency electric analyses. Components (X, Y, Z).
JT	X, Y, Z, SUM	Total measurable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM).
MRE		Magnetics Reynolds number.
VOLU		Volume of volume element.
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
FICT	TEMP	Fictive temperature.
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; I1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
FFLX	X, Y, Z	Fluid flux flow in poromechanics.
FGRA	X, Y, Z	Fluid pore pressure gradient in poromechanics.
PMSV	VRAT, PPRE, DSAT, RPER	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure-thermal elements.
YSIDX	TENS,SHEA	Yield surface activity status for Mohr-Coulomb, soil, concrete, and joint rock material models: 1 = yielded, 0 = not yielded.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Name Method		
Item	**Comp**	**Description**
FPIDX	TF01,SF01, TF02,SF02, TF03,SF03, TF04,SF04	Failure plane surface activity status for concrete and joint rock material models: 1 = yielded, 0 = not yielded. Tension and shear failure status are available for all four sets of failure planes.
NS	X, Y, Z, XY, YZ, XZ	Nominal strain for hyperelastic material, reported in the current configuration (unaffected by **RSYS (p. 1639)**).
MPLA	DMAC, DMAX	Microplane damage, macroscopic and maximum values.
MPDP	TOTA, TENS, COMP, RW	Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).
DAMAGE	1,2,3,MAX	Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).
GDMG		Damage
IDIS		Structural-thermal dissipation rate
BKS	X, Y, Z, XY, YZ, XZ	Total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements.
BKS1, . . . ,BKS5	X, Y, Z, XY, YZ, XZ	Superimposed components of the total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements when more than one superimposed back-stress component is defined.
EPFR		Free strain in porous media
FC1S	1,2,3,4,5,6	First set of six components of FCC crystal slip. Available for 3D elements only.
FC2S	1,2,3,4,5,6	Second set of six components of FCC crystal slip. Available for 3D elements only.
HC1S	1,2,3,4,5,6	Six components of HCP crystal slip on basal and prismatic systems. Available for 3D elements only.
HC2S	1,2,3,4,5,6	Six components of HCP crystal slip on pyramidal system. Available for 3D elements only.
HC3S	1,2,3,4,5,6	First set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC4S	1,2,3,4,5,6	Second set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC5S	1,2,3,4,5,6	Six components of HCP crystal slip on the second-order pyramidal system. Available for 3D elements only.
BC1S	1,2,3,4,5,6	First set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC2S	1,2,3,4,5,6	Second set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC3S	1,2,3,4,5,6	First set of six components of BCC slip on 112 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Name Method		
Item	**Comp**	**Description**
BC4S	1,2,3,4,5,6	Second set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC5S	1,2,3,4,5,6	First set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC6S	1,2,3,4,5,6	Second set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC7S	1,2,3,4,5,6	Third set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC8S	1,2,3,4,5,6	Fourth set of six components of BCC slip on 123 plane. Available for 3D elements only.
FC1H	1,2,3,4,5,6	First set of six components of FCC crystal hardness. Available for 3D elements only.
FC2H	1,2,3,4,5,6	Second set of six components of FCC crystal hardness. Available for 3D elements only.
HC1H	1,2,3,4,5,6	Sixcomponents of HCP crystal hardness on basal and prismatic systems. Available for 3D elements.
HC2H	1,2,3,4,5,6	Six components of HCP crystal hardness on pyramidal system. Available for 3D elements only.
HC3H	1,2,3,4,5,6	First set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC4H	1,2,3,4,5,6	Second set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC5H	1,2,3,4,5,6	Six components of HCP crystal hardness on the second-order pyramidal system. Available for 3D elements only.
BC1H	1,2,3,4,5,6	First set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC2H	1,2,3,4,5,6	Second set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC3H	1,2,3,4,5,6	First set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC4H	1,2,3,4,5,6	Second set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC5H	1,2,3,4,5,6	First set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC6H	1,2,3,4,5,6	Second set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC7H	1,2,3,4,5,6	Third set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC8H	1,2,3,4,5,6	Fourth set of six components of BCC hardness on 123 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Name Method		
Item	**Comp**	**Description**
XELG	1,2,3,45,6,EQV	Crystal Lagrangian strain in 11, 22, 33, 12, 23,13 directions and its equivalent. Available for 3D elements only.
SINT	RHO, ETA, SSTR, GRAIN	Sintering relative density, viscosity, sintering stress, and average grain size values.
Sequence Number Method		
Item	**Comp**	**Description**
SMISC	*snum*	Summable items.
NMISC	*snum*	Nonsummable items.
LS	*snum*	Line element elastic stresses.
LEPEL	*snum*	Line element strains.
LEPTH	*snum*	Line element thermal strains.
LEPPL	*snum*	Line element plastic strains.
LEPCR	*snum*	Line element creep strains.
LBFE	*snum*	Line element temperatures.

[a] The results for this postprocessing SEND component are invalid for ELBOW290 if that element is used with viscoelastic or viscohyperelastic materials.

[b] For more information about the meaning of contact status and its possible values, see Reviewing Results in POST1 in the *Contact Technology Guide*.

[c] For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . ., HTOP instead of HEAT.

[d] WEXT is calculated for element-based loading only (and not for nodal-force loading). WEXT is stored on elements to which loading has been applied; if surface elements are added on top of other elements, for example, and pressure loading is applied to the surface elements, WEXT is available for the surface elements only.

Table 112: ESOL - Selected Result (SRES) Component Labels

ESOL (p. 672),*NVAR*,*ELEM*,*NODE*,SRES,*Comp*	
Comp	**Description**
SVAR*n*	The *n*th state variable.
FLDUF0*n*	The *n*th user-defined field variable.
ItemComp (concatenated *Item* + *Comp* label) from Table 208: OSRESULT - Item and Component Labels (p. 1328)[a]	See the table (p. 1328) for the combined item and component description.

[a] See Example 10: Specifying and Retrieving Selected Result Output (p. 1329).

Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ESORT, *Item*, *Lab*, *ORDER*, *KABS*, *NUMB*

Sorts the element table.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item:

ETAB

(currently the only *Item* available)

Lab

element table label:

Lab

Any user-defined label from the **ETABLE (p. 687)** command (input in the *Lab* field of the **ETABLE (p. 687)** command).

ORDER

Order of sort operation:

0

Sort into descending order.

1

Sort into ascending order.

KABS

Absolute value key:

0

Sort according to real value.

1

Sort according to absolute value.

NUMB

Number of elements (element table rows) to be sorted in ascending or descending order (*ORDER*) before sort is stopped (remainder will be in unsorted sequence) (defaults to all elements).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Use ascending element number order.

Notes

The element table rows are sorted based on the column containing the *Lab* values. Use **EUSORT (p. 703)** to restore the original order. If **ESORT (p. 680)** is specified with PowerGraphics on (**/GRAPHICS (p. 866)**,POWER), then the nodal solution results listing (**PRNSOL (p. 1505)**) will be the same as with the full graphics mode (**/GRAPHICS (p. 866)**,FULL).

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Sort Elems

---

ESTIF, *KMULT*

Specifies the matrix multiplier for deactivated elements.

SOLUTION (p. 38): Birth and Death (p. 47)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KMULT

Stiffness matrix multiplier for deactivated elements (defaults to 1.0E-6).

Command Default

Use 1.0E-6 as the multiplier.

Notes

Specifies the stiffness matrix multiplier for elements deactivated with the **EKILL (p. 616)** command (birth and death).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Birth & Death>StiffnessMult

Main Menu>Solution>Load Step Opts>Other>Birth & Death>StiffnessMult

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ESURF, *XNODE*, *Tlab*, *Shape*

Generates elements overlaid on the free faces of selected nodes.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XNODE

Node number that is used only in the following two cases:

XNODE is a single extra node number (ID) used for generating SURF151 or SURF152 elements when KEYOPT(5)=1.

XNODE is a single pressure node number (ID) used for generating HSFLD241 or HSFLD242 elements.

There is no default. *XNODE* must be specified for the above cases. If *XNODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A parameter or parametric expression can be substituted for *XNODE*.

Tlab

Generates target, contact, and hydrostatic fluid elements with correct direction of normals.

This option is valid only with TARGE169, TARGE170, CONTA172, CONTA174, CONTA177, HSFLD241, and HSFLD242 elements.

TOP

Generates target and contact elements over beam and shell elements, or hydrostatic fluid elements over shell elements, with the normals the same as the underlying beam and shell elements (default).

BOTTOM

Generates target and contact elements over beam and shell elements, or hydrostatic fluid elements over shell elements, with the normals opposite to the underlying beam and shell elements.

If target or contact elements *and* hydrostatic fluid elements are defined on the same underlying shell elements, you only need to use this option once to orient the normals opposite to the underlying shell elements.

REVERSE

Reverses the direction of the normals on existing selected target elements, contact elements, and hydrostatic fluid elements.

If target or contact elements *and* hydrostatic fluid elements are defined on the same underlying shell elements, you only need to use this option once to reverse the normals for all selected elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shape

Used to specify the element shape for target element TARGE170 (*Shape* = LINE or POINT) or TARGE169 elements (*Shape* = POINT).

(blank)

The target element takes the same shape as the external surface of the underlying element (default).

LINE

Generates LINE or PARA (parabolic) segments on exterior of selected 3D elements.

POINT

Generates POINT segments on selected nodes.

Notes

The **ESURF (p. 682)** command generates elements of the currently active element type overlaid on the free faces of existing elements. For example, surface elements (such as SURF151, SURF152, SURF153, SURF154, or SURF159) can be generated over solid elements (such as PLANE55, SOLID70, PLANE182, SOLID185, or SOLID272, respectively).

Element faces are determined from the selected node set (**NSEL (p. 1266)**) and the load faces for that element type. The operation is similar to that used for generating element loads from selected nodes via the **SF (p. 1733)**,ALL command, except that elements (instead of loads) are generated. All nodes on the face must be selected for the face to be used. For shell elements, only face one of the element is available. If nodes are shared by adjacent selected element faces, the faces are not free and no element is generated.

Elements created by **ESURF (p. 682)** are oriented such that their surface load directions are consistent with those of the underlying elements. Carefully check generated elements and their orientations.

Generated elements use the existing nodes and the active **MAT (p. 1111)**, **TYPE (p. 2036)**, **REAL (p. 1579)**, and **ESYS (p. 685)** attributes. The exception is when *Tlab* = REVERSE. The reversed target and contact elements have the same attributes as the original elements. If the underlying elements are solid elements, *Tlab* = TOP or BOTTOM has no effect.

When the command generates a target element, the shape is by default the same as that of the underlying element. Issue **ESURF (p. 682)**,,,LINE or **ESURF (p. 682)**,,,POINT to generate LINE, PARA, and POINT segments.

The **ESURF (p. 682)** command can also generate the 2D or 3D node-to-surface element CONTA175, based on the selected node components of the underlying solid elements. When used to generate CONTA175 elements, all **ESURF (p. 682)** arguments are ignored. (If CONTA175 is the active element type, the path **Main Menu> Preprocessor> Modeling> Create> Elements> Node-to-Surf** uses **ESURF (p. 682)** to generate elements.)

To generate SURF151 or SURF152 elements that have two extra nodes from FLUID116 elements, KEYOPT(5) for SURF151 or SURF152 is first set to 0 and **ESURF (p. 682)** is issued. Then KEYOPT(5) for SURF151

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or SURF152 is set to 2 and **MSTOLE (p. 1200)** is issued. For more information, see Using the Surface Effect Elements in the *Thermal Analysis Guide*.

For hydrostatic fluid elements HSFLD241 and HSFLD242, the **ESURF (p. 682)** command generates triangular (2D) or pyramid-shaped (3D) elements with bases that are overlaid on the faces of selected 2D or 3D solid or shell elements. The single vertex for all generated elements is at the pressure node specified as *XNODE*. The generated elements fill the volume enclosed by the solid or shell elements. The nodes on the overlaid faces have translational degrees of freedom, while the pressure node shared by all generated elements has a single hydrostatic pressure degree of freedom, HDSP (see HSFLD241 and HSFLD242 for more information about the pressure node).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Inf Acoustic

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Node to Surf

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Generl Surface>Extra Node

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Generl Surface>No extra Node

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf to Surf

---

ESYM, - -, *NINC*, *IEL1*, *IEL2*, *IEINC*

Generates elements from a pattern by a symmetry reflection.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

--

Unused field.

NINC

Increment nodes in the given pattern by *NINC*.

IEL1*, *IEL2*, *IEINC

Reflect elements from pattern beginning with *IEL1* to *IEL2* (defaults to *IEL1*) in steps of *IEINC* (defaults to 1). If *IEL1* = ALL, *IEL2* and *IEINC* are ignored and pattern is all selected elements (**ESEL (p. 661)**). If *IEL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *IEL1* (*IEL2* and *IEINC* are ignored).

Notes

Generates additional elements from a given pattern (similar to **EGEN (p. 603)**) except with a symmetry reflection. The operation generates a new element by incrementing the nodes on the original element, and reversing and shifting the node connectivity pattern. For example, for a 4-node 2D element, the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nodes in positions I, J, K, and L of the original element are placed in positions J, I, L, and K of the reflected element.

Similar permutations occur for all other element types. For line elements, the nodes in positions I and J of the original element are placed in positions J and I of the reflected element.

It is recommended that symmetry elements be displayed and graphically reviewed.

If the nodes are also reflected (as with the **NSYM (p. 1289)** command), this pattern is such that the orientation of the symmetry element remains similar to the original element (that is, clockwise elements are generated from clockwise elements).

For a non-reflected node pattern, the reversed orientation has the effect of reversing the outward normal direction (clockwise elements are generated from counterclockwise elements).

Because nodes may be defined anywhere in the model independently of this command, any orientation of the symmetry elements is possible. See also the **ENSYM (p. 644)** command for modifying existing elements.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Elements>Auto Numbered

---

ESYS, *KCN*

Sets the element coordinate system attribute pointer.

PREP7 (p. 22): Meshing (p. 29)
PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Coordinate system number:

0

Use element coordinate system orientation as defined (either by default or by KEYOPT setting) for the element (default).

N

Use element coordinate system orientation based on local coordinate system *N* (where *N* must be greater than 10). For global system 0, 1, or 2, define a local system *N* parallel to appropriate system with the **LOCAL (p. 1059)** or **CS (p. 435)** command (for example, **LOCAL (p. 1059)**,11,1).

Command Default

Use element coordinate system orientation as defined (either by default or by KEYOPT setting) for the element (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

ESYS (p. 685) identifies the local coordinate system to be used to define the element coordinate system of subsequently defined elements.

The command is valid for area and volume elements only.

For non-layered volume elements, the local coordinate system *N* is assigned as the element coordinate system. *N* refers to the coordinate system reference number (*KCN*) defined via **LOCAL (p. 1059)** (or a similar command).

For shell and layered volume elements, the x and y axes of the local coordinate system *N* are projected onto the shell or layer plane to determine the element coordinate system.

For more information, see Understanding the Element Coordinate System.

Element coordinate system numbers can be displayed (**/PNUM (p. 1457)**).

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

ET, *ITYPE*, *Ename*, *KOP1*, *KOP2*, *KOP3*, *KOP4*, *KOP5*, *KOP6*, *INOPR*

Defines a local element type from the element library.

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITYPE

An arbitrary local element-type number. Defaults to 1 + current maximum.

Ename

A full element name (such as **pipe288**) *or* element number only (such as **288**), as given in the element library.

KOP1, KOP2, KOP3, . . . , KOP6

KEYOPT values (1 through 6) as desired for the specified element.

INOPR

Specify 1 to suppress all element-solution printout for this element type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **ET (p. 686)** command selects an element type from the element library and establishes it as a local element type for the current model. Information derived from the element type is used for subsequent commands, so the **ET (p. 686)** command(s) should be issued early.

Although specifying a full element name (category prefix and number such as **pipe288**) for *Ename* is optional, it is still displayed in the output.

Ename = 0 is a special option to ignore the specified element type during solution without removing it from the model. The option is valid only for the previously defined element type. (The preferred method for ignoring elements is to issue **ESEL (p. 661)** or another selection command (p. 14).)

KOPn are element key options (KEYOPT(*n*)), used to enable or disable options available for the given element. Available KEYOPT values are shown each element documented in the element library. If KEYOPT(7) and higher are needed, input those KEYOPT values values via **KEYOPT (p. 979)**.

Although **ET (p. 686)** defines an element type local to your model, you must point to the desired local element type before meshing. To do so, issue **TYPE (p. 2036)** (or a similar command such as **KATT (p. 969)**, **LATT (p. 1014)**, **AATT (p. 85)**, or **VATT (p. 2075)**).

Menu Paths

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

ETABLE, *Lab*, *Item*, *Comp*, *Option*

Fills a table of element values for further processing.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Any unique user-defined label for use in subsequent commands and output headings. A valid label has a maximum of eight characters and is not a general predefined *Item* label. Default: An eight-character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

If the same as a previous user label, the result item is included under the same label. Up to 200 different labels can be defined.

The following predefined labels are reserved:

REFL

Refills all tables previously defined with the **ETABLE (p. 687)** commands (not the **CALC (p. 283)** module commands) according to the latest **ETABLE (p. 687)** specifications. It is convenient for refilling tables after the load step (**SET (p. 1724)**) has changed. Remaining fields are ignored.

STAT

Displays stored table values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ERAS

Erases the entire table.

Item

Label identifying the item. General item labels are shown in the tables below. Some items also require a component label. Character parameters are valid. *Item* = ERAS erases a *Lab* column. For selected result output, specify SRES and see Table 114: ETABLE - Selected Result (SRES) Component Labels (p. 698).

Comp

Component of the item (if required). General component labels are shown in the tables below. Character parameters can be used.

Option

Option for storing element table data:

MIN

Store minimum element nodal value of the specified item component.

MAX

Store maximum element nodal value of the specified item component.

AVG

Store averaged element centroid value of the specified item component (default).

Notes

ETABLE (p. 687) defines a table of values per element (the element table) for use in further processing. The element table is organized similar to a spreadsheet, with rows representing all selected elements and columns consisting of result items which have been moved into the table (*Item*,*Comp*) via **ETABLE (p. 687)**. Each column of data is identified by a user-defined label (*Lab*) for listings and displays.

After entering the data into the element table, you are not limited to listing or displaying your data (**PLESOL (p. 1401)**, **PRESOL (p. 1481)**, etc.). You can also perform many types of operations on your data, such as adding or multiplying columns (**SADD (p. 1643)**, **SMULT (p. 1814)**), defining allowable stresses for safety calculations (**SALLOW (p. 1644)**), or multiplying one column by another (**SMULT (p. 1814)**). For more information, see Getting Started in the *Basic Analysis Guide*.

For reinforcing elements, this command displays the results of reinforcing member (individual reinforcing) selected via the **LAYER (p. 1015)**,*N* command (where *N* is a given reinforcing member). **LAYER (p. 1015)**,0 (default) or **LAYER (p. 1015)**,1 selects the first reinforcing member.

Various results data can be stored in the element table. For example, many items for an element are inherently single-valued (one value per element). The single-valued items include: SERR, SDSG, TERR, TDSG, SENE, TENE, KENE, ASENE, PSENE, AKENE, PKENE, DENE, WEXT, AENE, JHEAT, JS, VOLU, and CENT. All other items are multivalued (varying over the element, such that there is a different value at each node). Because only one value is stored in the element table per element, an average value (based on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the number of contributing nodes) is calculated for multivalued items. Exceptions to this averaging procedure are FMAG and all element force items, which represent the sum only of the contributing nodal values.

Two methods of data access can be used with the **ETABLE (p. 687)** command. The method you select depends upon the type of data that you want to store. Some results can be accessed via a generic label (Component Name method), while others require a label and number (Sequence Number method).

The component name method is used to access the general element data (that is, element data which is generally available to most element types or groups of element types). All of the single-valued items and some of the more general multivalued items are accessible with the Component Name method. Various element results depend on the calculation method and the selected results location (**AVPRIN (p. 210)**, **RSYS (p. 1639)**, **LAYER (p. 1015)**, **SHELL (p. 1784)**, and **ESEL (p. 661)**).

Although nodal data is readily available for listing and display (**PRNSOL (p. 1505)**, **PLNSOL (p. 1425)**) without using the element table, you can also use the Component Name method to enter these results into the element table for further "worksheet" manipulation. (See Getting Started in the *Basic Analysis Guide* for more information.) A listing of the general *Item* and *Comp* labels for the component name method is shown in Table 113: ETABLE - General Result Item and Component Labels (p. 690). See Table 114: ETABLE - Selected Result (SRES) Component Labels (p. 698) for a list of selected result (*Item* = SRES) *Comp* labels.

The sequence number method enables you to view results for data that is not averaged (such as pressures at nodes, temperatures at integration points, etc.), or data that is not easily described in a generic fashion (such as all derived data for structural line elements and contact elements, all derived data for thermal line elements, layer data for layered elements, etc.). A table illustrating the *Items* (such as LS, LEPEL, LEPTH, SMISC, NMISC, SURF, etc.) and corresponding sequence numbers for each element is shown in the Output Data section of each element description.

Some element table data are reported in the results coordinate system. These include all component results (for example, UX, UY, etc.; SX, SY, etc.). Line element solution quantities (LS, LEPEL, LEPTH, LEPPL, LEPCR, LEPTO, and LEPTT) for ELBOW290 are also reported in the results coordinate system if euler angles are saved. The solution writes component results in the database and on the results file in the solution coordinate system. When you issue the **ETABLE (p. 687)** command, these results are then transformed into the results coordinate system (**RSYS (p. 1639)**) before being stored in the element table. The default results coordinate system is global Cartesian (**RSYS (p. 1639)**,0). All other data are retrieved from the database and stored in the element table with no coordinate transformation.

To display the stored table values, issue the **PRETAB (p. 1490)**, **PLETAB (p. 1411)**, or **ETABLE (p. 687)**,STAT command. To erase the entire table, issue **ETABLE (p. 687)**,ERAS. To erase a *Lab* column, issue **ETABLE (p. 687)**,*Lab*,ERAS.

When the GUI is enabled, if a **Delete** operation in a Define Element Table Data dialog box writes this command to a log file (`Jobname.log` or `Jobname.lgw`), the program sets *Lab* = blank, *Item* = ERASE, and *Comp* = an integer number. In this case, the program has assigned a value of *Comp* that corresponds to the location of a chosen variable name in the dialog list. It is *not* intended that you type in such a location value for *Comp* in a session; however, a file that contains a GUI-generated **ETABLE (p. 687)** command of this form can be used for batch input or the **/INPUT (p. 948)** command.

The MIN and MAX options are not available for thermal elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element table data option (*Option*) is not available for all output items. See the table below for supported items.

Table 113: ETABLE (p. 687) - General Result Item and Component Labels

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
Valid Item Labels for Degree of Freedom Results		
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP[a]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
GFV1, GFV2, GFV3		Nonlocal field values 1, 2, and 3.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity.
A	X, Y, Z	X, Y, or Z magnetic vector potential.
CONC		Concentration.
CURR		Current.
EMF		Electromotive force drop.
Valid Item and Component Labels for Element Results		
S[b]	X, Y, Z, XY, YZ, XZ	Component stress.
	1, 2, 3	Principal stress.
	INT	Stress intensity.
	EQV	Equivalent stress.
EPEL[b]	X, Y, Z, XY, YZ, XZ	Component elastic strain.
	1, 2, 3	Principal elastic strain.
	INT	Elastic strain intensity.
	EQV	Elastic equivalent strain.
EPTH[b]	X, Y, Z, XY, YZ, XZ	Component thermal strain.
	1, 2, 3	Principal thermal strain.
	INT	Thermal strain intensity.
	EQV	Thermal equivalent strain.
EPDI[b]	X, Y, Z, XY, YZ, XZ	Component diffusion strain.
	1, 2, 3	Principal diffusion strain.
	EQV	Diffusion equivalent strain.
	INT	Diffusion strain intensity.
EPPL[b]	X, Y, Z, XY, YZ, XZ	Component plastic strain.
	1, 2, 3	Principal plastic strain.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
	INT	Plastic strain intensity.
	EQV	Plastic equivalent strain.
EPCR[b]	X, Y, Z, XY, YZ, XZ	Component creep strain.
	1, 2, 3	Principal creep strain.
	INT	Creep strain intensity.
	EQV	Creep equivalent strain.
EPSW[b]		Swelling strain.
EPTO[b]	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (*excluding* thermal) (EPEL + EPPL + EPCR).
	1, 2, 3	Principal total mechanical strain.
	INT	Total mechanical strain intensity.
	EQV	Total equivalent mechanical strain.
EPTT[b]	X, Y, Z, XY, YZ, XZ	Component total strain *including* thermal, diffusion, and swelling (EPEL + EPTH + EPPL + EPDI + EPCR + EPSW).
	1, 2, 3	Principal total strain.
	INT	Total strain intensity.
	EQV	Total equivalent strain.
NL[b]	SEPL	Equivalent stress (from stress-strain curve).
	SRAT	Stress state ratio.
	HPRES	Hydrostatic pressure.
	EPEQ	Accumulated equivalent plastic strain.
	CREQ	Accumulated equivalent creep strain.
	PSV	Plastic state variable.
	PLWK	Plastic work/volume.
SEND[b]	ELASTIC[c]	Elastic strain energy density. (For viscoelastic and sintering materials, the stored energy.)
	PLASTIC	Plastic strain energy density.
	CREEP	Creep strain energy density.
	DAMAGE	Damage strain energy density.
	VDAM[c]	Viscoelastic dissipation energy density.
	VREG	Visco-regularization strain energy density.
	DISS	Structural-thermal dissipation.
	ENTO	Total strain energy density (sum of ELASTIC, PLASTIC, and CREEP strain energy densities).
SVAR	1 to MAX	State variable.
CDM	DMG	Damage variable.
	LM	Maximum previous strain energy for virgin material.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
FAIL	MAX[a][d]	Maximum of all active failure criteria defined at the current location (**FCTYP (p. 740)**) .
	EMAX[a][d]	Maximum Strain Failure Criterion.
	SMAX[a][d]	Maximum Stress Failure Criterion.
	TWSI[a][d]	Tsai-Wu Strength Index Failure Criterion.
	TWSR[a][d]	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion.
	HFIB[a][d][e]	Hashin Fiber Failure Criterion.
	HMAT[a][d][e]	Hashin Matrix Failure Criterion.
	PFIB[a][d][e]	Puck Fiber Failure Criterion.
	PMAT[a][d][e]	Puck Matrix Failure Criterion.
	L3FB[a][d][e]	LaRc03 Fiber Failure Criterion.
	L3MT[a][d][e]	LaRc03 Matrix Failure Criterion.
	L4FB[a][d][e]	LaRc04 Fiber Failure Criterion.
	L4MT[a][d][e]	LaRc04 Matrix Failure Criterion.
	USR1, USR2, ..., USR9[a][d][e][f]	User-defined failure criteria.
PFC	MAX[g]	Maximum of all failure criteria defined at current location.
	FT[g]	Fiber tensile failure criteria.
	FC[g]	Fiber compressive failure criteria.
	MT[g]	Matrix tensile failure criteria.
	MC[g]	Matrix compressive failure criteria.
PDMG	STAT	Damage status (0 = undamaged, 1 = damaged, 2 = completely damaged).
	FT	Fiber tensile damage variable.
	FC	Fiber compressive damage variable .
	MT	Matrix tensile damage variable .
	MC	Matrix compressive damage variable .
	S	Shear damage variable (S).
	SED	Energy dissipated per unit volume .
	SEDV	Energy per unit volume due to viscous damping.
FCMX[a][d]	LAY	Layer number where the maximum of all active failure criteria over the entire element occurs.
	FC	Number of the maximum-failure criterion over the entire element: 1 - EMAX 2 - SMAX 3 - TWSI

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
		4 - TWSR 5 - PFIB 6 - PMAT 7 - HFIB 8 - HMAT 9 - L3FB 10 - L3MT 11 - L4FB 12 - L4MT 13~21 - USR1~USR9
	VAL	Value of the maximum failure criterion over the entire element.
TG[h]	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF[h]	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum.
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum.
FMAG[f]	X, Y, Z, SUM	Component electromagnetic forces or vector sum.
SERR[e]		Structural error energy.
SDSG[e]		Absolute value of maximum variation of any nodal stress component.
TERR[e]		Thermal error energy.
TDSG[e]		Absolute value of the maximum variation of any nodal thermal gradient component.
F	X, Y, Z	Component structural force. Sum of element nodal values.
M	X, Y, Z	Component structural moment. Sum of element nodal values.
HEAT		Heat flow. Sum of element nodal values.
FLOW		Fluid flow. Sum of element nodal values.
AMPS		Current flow. Sum of element nodal values.
FLUX		Magnetic flux. Sum of element nodal values.
CSG	X, Y, Z	Component magnetic current segment.
RATE		Diffusion flow rate. Sum of element nodal values.
SENE		"Stiffness" energy or thermal heat dissipation (applies to all elements where meaningful). Same as TENE.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
AENE		Artificial energy of the element. This includes the sum of hourglass control energy and energy generated by in-plane drilling stiffness from shell elements (applies to all elements where meaningful). It also includes artificial energy due to contact stabilization. The energy is used for comparisons to SENE energy to predict the solution error due to artificial stiffness.
TENE		Thermal heat dissipation or "stiffness" energy (applies to all elements where meaningful). Same as SENE.
KENE		Kinetic energy (applies to all elements where meaningful).
ASENE		Amplitude "stiffness" energy.
PSENE		Peak "stiffness" energy.
AKENE		Amplitude kinetic energy.
PKENE		Peak kinetic energy.
DENE		Damping energy.
WEXT[i]		Work due to external load.
STEN		Elemental energy dissipation due to stabilization.
JHEAT		Element Joule heat generation.
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM).
JT	X, Y, Z, SUM	Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM).
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).
MRE		Magnetics Reynolds number.
VOLU		Element volume. Based on unit thickness for 2D plane elements (unless the thickness option is used) and on the full 360 degrees for 2D axisymmetric elements.
CENT	X, Y, Z	Undeformed X, Y, or Z location (based on shape function) of the element centroid in the active coordinate system.
BFE[b]	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
SMISC	*snum*	Element summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each applicable element description).
NMISC	*snum*	Element non-summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each applicable element description).
SURF	*snum*	Element surface data value at sequence number *snum*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
CONT	STAT[d]	Contact status: 3 = closed and sticking 2 = closed and sliding 1 = open but near contact 0 = open and not near contact
	PENE	Contact penetration (zero or positive).
	PRES	Contact pressure.
	SFRIC	Contact friction stress.
	STOT	Contact total stress (pressure plus friction).
	SLIDE	Contact sliding distance.
	GAP	Contact gap distance (0 or negative).
	FLUX	Total heat flux at contact surface.
	CNOS	Total number of contact status changes during substep.
	FPRS	Fluid penetration pressure.
TOPO		Densities used for topological optimization.
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; I1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
ESIG[b]	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
	1, 2, 3	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
	GPOR	Porosity due to void growth.
	NPOR	Porosity due to void nucleation.
FFLX	X,Y,Z	Fluid flow flux in poromechanics.
FGRA	X,Y,Z	Fluid pore pressure gradient in poromechanics.
FICT[b]	TEMP	Fictive temperature.
PMSV	VRAT, PPRE, DSAT, RPER	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure-thermal elements.
YSIDX	TENS,SHEA	Yield surface activity status for Mohr-Coulomb, soil, concrete, and joint rock material models: 1 = yielded, 0 = not yielded.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
FPIDX	TF01,SF01, TF02,SF02, TF03,SF03, TF04,SF04	Failure plane surface activity status for concrete and joint rock material models: 1 = yielded, 0 = not yielded. Tension and shear failure status are available for all four sets of failure planes.
NS	X, Y, Z, XY, YZ, XZ	Nominal strain for hyperelastic material, reported in the current configuration (unaffected by **RSYS (p. 1639)**).
MPLA	DMAC, DMAX	Microplane damage, macroscopic and maximum values.
MPDP	TOTA, TENS, COMP, RW	Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).
DAMAGE	1,2,3,MAX	Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).
GDMG		Damage
IDIS		Structural-thermal dissipation rate
BKS	X, Y, Z, XY, YZ, XZ	Total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements.
BKS1, . . . , BKS5	X, Y, Z, XY, YZ, XZ	Superimposed components of the total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements when more than one superimposed back-stress component is defined.
EPFR		Free strain in porous media
FC1S	1,2,3,4,5,6	First set of six components of FCC crystal slip. Available for 3D elements only.
FC2S	1,2,3,4,5,6	Second set of six components of FCC crystal slip. Available for 3D elements only.
HC1S	1,2,3,4,5,6	Six components of HCP crystal slip on basal and prismatic systems. Available for 3D elements only.
HC2S	1,2,3,4,5,6	Six components of HCP crystal slip on pyramidal system. Available for 3D elements only.
HC3S	1,2,3,4,5,6	First set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC4S	1,2,3,4,5,6	Second set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC5S	1,2,3,4,5,6	Six components of HCP crystal slip on the second-order pyramidal system. Available for 3D elements only.
BC1S	1,2,3,4,5,6	First set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC2S	1,2,3,4,5,6	Second set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC3S	1,2,3,4,5,6	First set of six components of BCC slip on 112 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
BC4S	1,2,3,4,5,6	Second set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC5S	1,2,3,4,5,6	First set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC6S	1,2,3,4,5,6	Second set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC7S	1,2,3,4,5,6	Third set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC8S	1,2,3,4,5,6	Fourth set of six components of BCC slip on 123 plane. Available for 3D elements only.
FC1H	1,2,3,4,5,6	First set of six components of FCC crystal hardness. Available for 3D elements only.
FC2H	1,2,3,4,5,6	Second set of six components of FCC crystal hardness. Available for 3D elements only.
HC1H	1,2,3,4,5,6	Sixcomponents of HCP crystal hardness on basal and prismatic systems. Available for 3D elements.
HC2H	1,2,3,4,5,6	Six components of HCP crystal hardness on pyramidal system. Available for 3D elements only.
HC3H	1,2,3,4,5,6	First set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC4H	1,2,3,4,5,6	Second set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC5H	1,2,3,4,5,6	Six components of HCP crystal hardness on the second-order pyramidal system. Available for 3D elements only.
BC1H	1,2,3,4,5,6	First set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC2H	1,2,3,4,5,6	Second set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC3H	1,2,3,4,5,6	First set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC4H	1,2,3,4,5,6	Second set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC5H	1,2,3,4,5,6	First set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC6H	1,2,3,4,5,6	Second set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC7H	1,2,3,4,5,6	Third set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC8H	1,2,3,4,5,6	Fourth set of six components of BCC hardness on 123 plane. Available for 3D elements only.
XELG	1,2,3,45,6,EQV	Crystal Lagrangian strain in 11, 22, 33, 12, 23,13 directions and its equivalent. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels ETABLE (p. 687), *Lab, Item, Comp*		
Item	**Comp**	**Description**
SINT	RHO, ETA, SSTR, GRAIN	Sintering relative density, viscosity, sintering stress, and average grain size values.

[a] For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP.

[b] Element table option (*Option*) is available for this element output data item.

[c] The results for this postprocessing SEND component are invalid for ELBOW290 when that element is used with viscoelastic or viscohyperelastic materials.

[d] For MPC-based contact definitions, the value of STAT can be negative. This indicates that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.

[e] Some element- and material-type limitations apply. For more information, see **PRERR (p. 1480)**.

[f] When using the **EMFT (p. 624)** procedure to calculate electromagnetic force (PLANE121, SOLID122, SOLID123, PLANE233, SOLID236 or SOLID237 elements only), the FMAG sum will be zero or near zero.

[g] Failure criteria are based on the effective stresses in the damaged material.

[h] *Comp* = SUM is not supported for coupled pore-pressure-thermal (CPT*nnn*) elements.

[i] WEXT is calculated for element-based loading only (and not for nodal-force loading). WEXT is stored on elements to which loading has been applied; if surface elements are added on top of other elements, for example, and pressure loading is applied to the surface elements, WEXT is available for the surface elements only.

Table 114: ETABLE - Selected Result (SRES) Component Labels

ETABLE (p. 687),*Lab*,SRES,*Comp*	
Comp	**Description**
SVAR*n*	The *n*th state variable.
FLDUF0*n*	The *n*th user-defined field variable.
ItemComp (concatenated *Item* + *Comp* label) from Table 208: OSRESULT - Item and Component Labels (p. 1328)[a]	See the table (p. 1328) for the combined item and component description.

[a] See Example 10: Specifying and Retrieving Selected Result Output (p. 1329).

Menu Paths

Main Menu>General Postproc>Element Table>Define Table

Main Menu>General Postproc>Element Table>Erase Table

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ETCHG, *Cnv*

Changes element types to their corresponding types.

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Cnv

Converts the element types to the corresponding type. Valid labels are:

TTS

Thermal to Structural

STT

Structural to Thermal

MTT

Magnetic to Thermal

FTS

Fluid to Structural

ETS

Electrostatic to Structural

ETT

Electrical to Thermal

Notes

Changes the currently defined element types to their corresponding types. Elements without a companion element (listed above) are not switched and should be switched with the **ET (p. 686)** command to an appropriate element type or to a null element. The KEYOPT values for the switched element types are reset to zero or to their default values. You must check these values to see if they are still meaningful. Additionally, if *Cnv* = ETI, ITE, or TTE, all real constants are set to zero.

If *Cnv* = ITE, you will need to choose a material model that corresponds to your previously-defined material properties. If working interactively, the application prompts you to do so.

Element Pairs

TTS -- Thermal to Structural

33 > 180	67 > 42	75 > 25	131 > 181
55 > 182	69 > 45	77 > 183	151 > 153

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

70 > 185
71 > 21
78 > 83
87 > 187
90 > 186
152 > 154
155 > 156
278 > 185
279 > 186
291 > 187
292 > 182
293 > 183

STT -- Structural to Thermal

21 > 71
25 > 75
42 > 55
45 > 70
56 > 55
58 > 70
62 > 70
64 > 70
73 > 70
74 > 77
82 > 77
83 > 78
84 > 55/77
86 > 70
88 > 77
89 > 90
91 > 132
92 > 87
93 > 132
95 > 90
106 > 55
107 > 70
108 > 77
153 > 151
154 > 152
156 > 155
158 > 87
180 > 33
181 > 131
182 > 55
183 > 77
185 > 70
186 > 90
187 > 87

MTT -- Magnetic to Thermal

233 > 77
62 > 70
96 > 70
236 > 90
237 > 87

ETS -- Electrostatic to Structural

121 > 182/183
122 > 185/186
123 > 187

ETT -- Electrical to Thermal

230 > 77
231 > 90
232 > 87

Menu Paths

Main Menu>Preprocessor>Element Type>Switch Elem Type

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ETCONTROL, *Eltech*, *Eldegene*

Control the element technologies used in element formulation (for applicable elements).

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Eltech

Element technology control:

SUGGESTION

The program offers a suggestion for the best element technology before solving. If necessary, mixed u-P (KEYOPT(6)) is also included and reset. This behavior is the default.

SET

The program informs you of the best settings and resets any applicable KEYOPT settings automatically. This action overrides any previous manual settings.

OFF

Deactivates automatic selection of element technology. No suggestions are issued, and no automatic resetting occurs.

Eldegene

Element degenerated shape control:

ON

If element shapes are degenerated, the degenerated shape function is employed and enhanced strain, simplified enhanced strain, and B-bar formulations are turned off (default).

OFF

If element shapes are degenerated, regular shape functions are still used, and the specified element technologies (for example, enhanced strain, B-bar, uniform reduced integration) are still used.

Command Default

The program makes suggestions for the best element technology by default (**ETCONTROL (p. 701)**,SUGGESTION,ON).

Notes

This command is valid for elements SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SHELL208, SHELL209, PLANE222, PLANE223, SOLID225, SOLID226, SOLID227, REINF264, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information, see Automatic Selection of Element Technologies and Formulations in the *Element Reference*.

Menu Paths

Main Menu>Preprocessor>Element Type>Elem Tech Control

ETDELE, *ITYP1*, *ITYP2*, *INC*

Deletes element types.

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITYP1*, *ITYP2*, *INC

Deletes element types from *ITYP1* to *ITYP2* (defaults to *ITYP1*) in steps of *INC* (defaults to 1). If *ITYP1* = ALL, *ITYP2* and *INC* are ignored and all element types are deleted. Element types are defined with the **ET (p. 686)** command.

Menu Paths

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

ETLIST, *ITYP1*, *ITYP2*, *INC*

Lists currently defined element types.

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITYP1*, *ITYP2*, *INC

Lists element types from *ITYP1* to *ITYP2* (defaults to *ITYP1*) in steps of *INC* (defaults to 1). If *ITYP1* = ALL (default), *ITYP2* and *INC* are ignored and all element types are listed.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Properties>Element Types

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ETYPE

Specifies "Element types" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Element Types

EUSORT

Restores original order of the element table.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Changing the selected element set (**ESEL (p. 661)**) also restores the original element order.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Unsort Elems

EWRITE, *Fname*, *Ext*, --, *KAPPND*, *Format*

Writes elements to a file.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (up to 248 characters, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to ELEM if *Fname* is blank.

--

Unused field.

KAPPND

Append key:

0

Rewind file before the write operation.

1

Append data to the end of the existing file.

Format

Format key:

SHORT

I6 format (default).

LONG

I8 format. Switches automatically to I10 if entity IDs are large.

Notes

Writes the selected elements to a file. The write operation is unnecessary in a standard Mechanical APDL run but is provided as convenience to users wanting a coded element file.

If issuing **EWRITE (p. 703)** from Mechanical APDL to be used in Mechanical APDL, issue **NWRITE (p. 1305)** to store nodal information for later use.

Only elements having all of their nodes defined (and selected) are written. Data are written in a coded format. The data description of each record is: I, J, K, L, M, N, O, P, MAT, TYPE, REAL, SECNUM, ESYS, IEL, where MAT, TYPE, REAL, and ESYS are attribute numbers, SECNUM is the beam section number, and IEL is the element number.

The format is (14I6) if *Format* = SHORT, and (14I8 or 14I10) if *Format* = LONG.

One element description per record is written for elements having <= 8 nodes. For elements having > 8 nodes, nodes 9 and above are written on a second record using the same format.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Write Elem File

---

EXBOPT,*OUTINV2*,*OUTTCMS*,*OUTSUB*,*OUTCMS*,*OUTCOMP*,*OUTRM*,*NOINV*,*OUTELE*

Specifies *.exb* file output options in a CMS generation pass.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

OUTINV2

Output control for 2nd order invariant:

0

Do not output (default).

1

Output the second order invariant.

OUTTCMS

Output control for `.tcms` file:

0

Do not output (default).

1

Output the `.tcms` file.

OUTSUB

Output control for `.sub` file:

0

Do not output (default).

1

Output the `.sub` file.

OUTCMS

Output control for `.cms` file:

0

Do not output (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Output the `.cms` file.

OUTCOMP

Output control for node and element component information:

0

Do not output any component information.

1

Output node component information only.

2

Output element component information only.

3

Output both node and element component information (default).

OUTRM

Output control for the recovery matrix:

0

Do not output (default).

1

Output the recovery matrix to `file.exb`.

2

Output the recovery matrix to a separate file, `file_RECOVER.exb`.

NOINV

Invariant calculation:

0

Calculate all invariants (default).

1

Suppress calculation of the 1st and 2nd order invariants. NOINV = 1 suppresses OUTINV2 = 1.

OUTELE

Output control for the element data:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Do not output (default).

1

Output the element data.

Command Default

Default settings as described for each argument are used.

Notes

When the body property file (`file.exb`) is requested in a CMS generation pass (**CMSOPT (p. 378)**,,,,,,EXB command), the `.tcms`, `.sub`, and `.cms` files are not output by default. Use the **EXBOPT (p. 705)** command to request these files, as needed.

EXBOPT (p. 705) can also be used to manage some content in the `.exb` file for improving performance and storage (see the *OUTINV2*, *OUTCOMP*, *OUTRM*, *NOINV*, and *OUTELE* arguments described above).

If both recovery matrix output (*OUTRM* = 1 or 2) and the `.tcms` file (*OUTTCMS* = 1) are requested, the `.tcms` file writing is turned off due to potentially large in-core memory use.

For more information on how to generate `file.exb`, see Ansys Interface to AVL EXCITE in the *Substructuring Analysis Guide*

Menu Paths

This command cannot be accessed from a menu.

---

*EXIT

Exits a do-loop.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

The command following the ***ENDDO (p. 636)** is executed next. The exit option may also be conditional [Use the ***IF (p. 927)**]. The ***EXIT (p. 707)** command must appear on the same file as the ***DO (p. 567)** command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/EXIT,*Slab*,*Fname*,*Ext*,--

Stops the run and returns control to the system.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Slab

Mode for saving the database:

MODEL

Save the model data (solid model, finite element model, loadings, etc.) only (default).

SOLU

Save the model data and the solution data (nodal and element results).

ALL

Save the model data, solution data and post data (element tables, path results, etc.)

NOSAVE

Do not save any data on File.db (an existing DB file will not be overwritten).

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name, defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to DB if *Fname* is blank.

--

Unused field.

Notes

The current database information may be written on File.db or a named file. If File.db already exists, a backup file (File.dbb) will also be written whenever a new File.db is written.

This command is valid in any processor. Issuing this command at any point will exit the program.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>File>Exit

EXOPTION, *Ldtype*, *Option*, *VALUE*

Specifies the EXPROFILE (p. 719) options for the Mechanical APDL to Ansys CFX or Ansys Fluent profile file transfer.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ldtype

Load type:

SURF

Surface load

VOLU

Volume load

Option

Surface options:

Precision

Number of significant digits for the fractional part of real data

Connectivity

Key to include face connectivity in the exported profile file

Volume options:

Precision

Number of significant digits after the decimal for real data

VALUE

Specify the value for either Precision or Connectivity.

For Precision, specify the number of significant digits. Can be any value between 1 to 20, default 8. When 0 or an invalid value is specified, the program will use the default value of 8 and issue a warning message.

For Connectivity, specify the key to include the element face connectivity data for surface loads (does not support volume loads):

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Do not include the connectivity data in the exported file (default)

ON

Include the connectivity data in the exported file

Menu Paths

This command cannot be accessed from a menu.

EXP,*IR*,*IA*,--,--,*Name*,--,--,*FACTA*,*FACTB*

Forms the exponential of a variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

--,--

Unused fields.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--,--

Unused fields.

FACTA

Scaling factor applied to variable *IA* (defaults to 1.0).

FACTB

Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Forms the exponential of a variable according to the operation:

IR = *FACTB**EXP(*FACTA* x *IA*)

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Exponentiate

EXPAND,*Nrepeat*,MODAL,*HIndex*,*Icsys*,*SctAng*,*--*,*Phase*

Displays the results of a modal cyclic symmetry analysis.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Nrepeat

Number of sector repetitions for expansion. The default is 0 (no expansion).

MODAL

Specifies that the expansion is for a modal cyclic symmetry analysis.

HIndex

The harmonic index ID for the results to expand.

Icsys

The coordinate system number used in the modal cyclic symmetry solution. The default is the global cylindrical coordinate system (specified via the **CSYS (p. 441)** command where *KCN* = 1).

SctAng

The sector angle in degrees, equal to 360 divided by the number of cyclic sectors.

--

This field is reserved for future use.

Phase

The phase angle in degrees to use for the expansion. The default is 0. Typically, the value is the peak displacement (or stress/strain) phase angle obtained via the **CYCPHASE (p. 472)** command.

Notes

Issue this command to display the results of a modal cyclic symmetry analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you issue the **EXPAND (p. 711)**,*Nrepeat* command, subsequent **SET (p. 1724)** commands read data from the results file and expand them to *Nrepeat* sectors. As long as no entities have been modified, this expansion can be negated (that is, reverted to single sector) by issuing **EXPAND (p. 711)** with no arguments. If you modify entities and wish to return to the partial model, use the Session Editor (see Restoring Database Contents in the *Operations Guide*).

EXPAND (p. 711) displays the results and allows you to print them, as if for a full model. The harmonic index (automatically retrieved from the results file) appears in the legend column.

When plotting or printing element strain energy (SENE), the **EXPAND (p. 711)** command works with brick or tet models only. Element kinetic energy (KENE) plotting or printing is not supported.

EXPAND (p. 711) is a specification command valid only in POST1. It is significantly different from the **/CYCEXPAND (p. 454)** command in several respects, (although you can use either command to display the results of a modal cyclic symmetry analysis):

- **EXPAND (p. 711)** has none of the limitations of the **/CYCEXPAND (p. 454)** command.
- **EXPAND (p. 711)** changes the database by modifying the geometry, the nodal displacements, and element stresses as they are read from the results file, whereas the **/CYCEXPAND (p. 454)** command does not change the database.

Caution:

The **EXPAND (p. 711)** command creates new nodes and elements; therefore, saving (or issuing the **/EXIT (p. 708)**, ALL command) after issuing the **EXPAND (p. 711)** command can result in large databases.

Menu Paths

Main Menu>General Postproc>Cyc Expansion

/**EXPAND**,*Nrepeat1*,*Type1*,*Method1*,*DX1*,*DY1*,*DZ1*,*Nrepeat2*,*Type2*,*Method2*,*DX2*,*DY2*,*DZ2*,*Nrepeat3*,*Type3*,*Method3*,*DX3*,*DY3*,*DZ3*

Allows the creation of a larger graphic display than represented by the actual finite element analysis model.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Nrepeat1, Nrepeat2, Nrepeat3

The number of repetitions required for the element pattern. The default is 0 (no expansion).

Type1, Type2, Type3

The type of expansion requested.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RECT

Causes a Cartesian transformation of DX, DY, and DZ for each pattern (default).

POLAR

Causes a polar transformation of DR, D-Theta and DZ for each pattern.

AXIS

Causes 2D axisymmetric expansion (that is, rotates a 2D model created in the X-Y plane about the Y axis to create a 3D model).

LRECT

Causes a Cartesian transformation of DX, DY, and DZ for each pattern about the current local coordinate system (specified via the **CSYS (p. 441)** command).

LPOLAR

Causes a polar transformation of DR, D-Theta, and DZ for each pattern about the local coordinate system (specified via the **CSYS (p. 441)** command).

Method1*, *Method2*, *Method3

The method by which the pattern is repeated.

FULL

Causes a normal repeat of the pattern (default).

HALF

Uses a symmetry transformation for alternate repeats (to produce an image of a complete circular gear from the image of half a tooth, for example).

DX1*, *DY1*, *DZ1*, *DX2*, *DY2*, *DZ2*, *DX3*, *DY3*, *DZ3

The Cartesian or polar increments between the repeated patterns. Also determines the reflection plane. Reflection is about the plane defined by the normal vector (DX, DY, DZ). If you want no translation, specify a small nonzero value. For a half-image expansion, the increment DX, DY, or DZ is doubled so that POLAR,HALF, ,45 produces full images on 90° centers, and RECT,HALF, ,1 produces full images on 2-meter centers.

Notes

You can use the **/EXPAND (p. 712)** command to perform up to three symmetry expansions at once (that is, X, Y, and Z which is equal to going from a 1/8 model to a full model). Polar expansions allow you to expand a wheel section into a half wheel, then into the half section, and then into the whole.

The command displays elements/results when you issue the **EPLOT (p. 648)** command or postprocessing commands.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The command works on all element and result displays, except as noted below. As the graphic display is created, the elements (and results) are repeated as many times as necessary, expanding the geometry and, if necessary, the displacements and stresses.

Derived results are not supported.

The **/EXPAND (p. 712)** command has the following limitations:

- It does not support solid model entities.
- POLAR, FULL or HALF operations are meaningful only in global cylindrical systems and are unaffected by the **RSYS (p. 1639)** or **DSYS (p. 582)** commands. Cartesian symmetry or unsymmetric operations also occur about the global Cartesian system.
- It does not average nodal results across sector boundaries, even for averaged plots (such as those obtained via the **PLNSOL (p. 1425)** command).
- Axisymmetric harmonic element results are not supported for *Type* = AXIS.

The **/EXPAND (p. 712)** command differs significantly from the **EXPAND (p. 711)** command in several respects:

- The uses of **/EXPAND (p. 712)** are of a more general nature, whereas the **EXPAND (p. 711)** command is intended *primarily* to expand modal cyclic symmetry results.
- **/EXPAND (p. 712)** does not change the database as does the **EXPAND (p. 711)** command.
- You cannot print results displayed via **/EXPAND (p. 712)**.

Menu Paths

Utility Menu>PlotCtrls>Style>Symmetry Expansion>

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Expansion by values

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Modal Cyclic Symmetry

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Periodic/Cyclic Symmetry Expansion

---

EXPASS,*Key*,--,--,*KeyStat*

Specifies an expansion pass of an analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Expansion pass key:

OFF

No expansion pass will be performed (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON

An expansion pass will be performed.

- - , - -

Unused fields.

KeyStat

Static correction vectors key:

ON

Include static correction vectors in the expanded displacements (default).

OFF

Do not include static correction vectors in the expanded displacements.

Notes

Specifies that an expansion pass of a modal, substructure, buckling, transient, or harmonic analysis is to be performed.

Note:

This separate solution pass requires an explicit **FINISH (p. 758)** from the preceding analysis and reentry into SOLUTION.

The *KeyStat* argument is applicable to the expansion pass of a substructure analysis, and to the expansion pass of a component mode synthesis (CMS) analysis when the CMS method is fixed-interface or free-interface. For a substructure analysis, the static correction vectors are the first terms of Equation 15.134 in the *Mechanical APDL Theory Reference*. For a CMS analysis, the static correction vectors are the third terms of Equation 15.158.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>ExpansionPass

Main Menu>Solution>Analysis Type>ExpansionPass

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*EXPORT,*Matrix*,*Format*,*Fname*,*Val1*,*Val2*,*Val3*

Exports a matrix to a file in the specified format.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Matrix

Name of the matrix to export (must be a matrix previously created with ***DMAT (p. 551)** or ***SMAT (p. 1801)**, or a vector previously created with ***VEC (p. 2085)**).

Format

Format of the output file:

MMF --

Export the matrix in the Matrix Market Format.

SUB --

Export the matrix in the SUB file format.

HBMAT --

Export the matrix in the Harwell-Boeing file format.

MAT --

Export the matrix in a native format, to be re-imported using the ***DMAT (p. 551)** or ***SMAT (p. 1801)** command.

EMAT --

Export the matrix to an existing EMAT file.

APDL --

Export the matrix to an APDL array parameter.

PS --

Export the matrix profile to a Postscript file.

DMIG --

Export the matrix in the DMIG file format.

CSV --

Export the matrix to an ASCII CSV (comma-separated values) file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fname

Name of the file (case-sensitive, 32-character maximum), or name of the array parameter if *Format* = APDL (no default).

Val1, *Val2*, *Val3*

Additional input. The meaning of *Val1* through *Val3* will vary depending on the specified *Format*. See table below for details.

Additional input for *Format* = SUB, HBMAT, EMAT, APDL, PS, DMIG, and CSV				
Format	***Val1***	***Val2***	***Val3***	**Description**
SUB	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix RHS - Load vector	Matrix containing the row information (see Notes (p. 718)). Not needed if the matrix was imported from an existing SUB or DMIG file.	WAIT DONE	Identify matrix type to write and specify when to perform the export (see Notes (p. 718)).
HBMAT	File format: ASCII BINARY	Matrix type (optional; see Notes (p. 718)): STIFF - Stiffness matrix MASS - Mass matrix DAMP - Damping matrix	(not used)	Identify the output format and matrix type to write (see Notes (p. 718)).
EMAT	Matrix type: STIFF - Stiffness (or conductivity) matrix MASS - Mass (or specific heat) matrix DAMP - Damping matrix	Element number	(not used)	Identify the matrix type and the element number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	STRESS - Stress-stiffness matrix			
APDL	First column	Last column	(not used)	Block of columns to export.
PS	Color key: BW - black and white (default) COLOR - color	(not used)	(not used)	Identify the output format.
DMIG	Matrix containing the row information (see Notes (p. 718)). Not needed if the matrix was imported from an existing SUB or DMIG file.	File format: FREE - Free field format LARGE - Large field format (default)	(not used)	The row information vector specifies the mapping between row number, node, and DOF number.
CSV	Number of decimal places (default = 10)	Delimiter character (default=','). Use 'S' for space, 'T' for tab delimiter.	(not used)	Identify the accuracy and formatting.

Notes

Only sparse matrices can be exported to Postscript files. This option plots the matrix profile as a series of dots.

If you want to create a `.sub` file from several matrices, you need to set *Val3* = WAIT for all matrices but the last, and *Val3* = DONE for the last one. The export will be effective at the last ***EXPORT (p. 716)** command.

To create a `.sub` file or `.dmig` file from scratch, you must supply the row information array. (Specify this array in the *Val2* field for `.sub` or in the *Val1* field for `.dmig`.) This must be an *m* x 2 array, where *m* is the size of the matrix. The first column is the node number and the second column is the DOF number corresponding to each row of the matrix.

When exporting an HBMAT file in ASCII format, you can include the matrix type in the header of the file by specifying the matrix type in the *Val2* field. The matrix type is not included in the header if *Val2* is empty. If *Val1* = BINARY, *Val2* is not used.

The ***EXPORT (p. 716)** command is not applicable to sparse matrices initialized from `.full` files by means of the NOD2SOLV option on the ***SMAT (p. 1801)** command (that is, ***SMAT (p. 1801)**,,,IMPORT,FULL,,NOD2SOLV).

The `.csv` file format does not support sparse matrices.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

EXPROFILE, *Ldtype*, *Load*, *VALUE*, *Pname*, *Fname*, *Fext*, *Fdir*

Exports Mechanical APDL interface data on selected nodes to an Ansys CFX or Ansys Fluent Profile file.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ldtype

Load type:

SURF

Surface load.

VOLU

Volumetric load.

Load

Surface loads:

DISP

Displacement (in a static analysis) or mode shape and global parameters (in a modal analysis).

MODE

Normalized mode shape and global parameters (in a modal analysis only).

TEMP

Temperature.

HFLU

Heat flux.

Volumetric loads:

DISP

Displacement.

FORC

Force.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HGEN

Heat generation.

VALUE

If a positive integer, specifies the number of the surface or volume interface. If zero (default), the selected nodes or Named Selection are used.

Pname

Field name in CFX Profile file (32-character maximum). Defaults to `jobname_bcploadnumber` for a surface load and `jobname_subdloadnumber` for volumetric load.

Fname

The CFX Profile filename (248-character maximum). Defaults to `jobname_bcploadnumber` for a surface load and `jobname_subdloadnumber` for a volumetric load.

Fext

The Profile file extension (8-character maximum). Defaults to `.csv`.

Fdir

The Profile file directory (248-character maximum). Defaults to current directory.

Notes

By default, the **EXPROFILE (p. 719)** command assumes the data it writes to the Profile file are in SI units. For models not described in SI units, issue the **EXUNIT (p. 725)** command as needed to write the correct unit labels on the Profile file.

For a modal analysis, if *Load* = DISP or MODE, global parameters including mass, frequency, and maximum displacement are also written to the Ansys CFX Profile file. You should therefore issue the **EXUNIT (p. 725)** command for DISP, TIME, and MASS.

Verify that the coordinate system is set to the global Cartesian (**RSYS (p. 1639)**,0) before using this command.

To transfer multiple loads across an interface, specify a unique file name and extension for each load.

Force (FORC) and heat generation (HGEN) are per-unit volume.

For modal analysis, this command will write global parameters including mass, frequency, and maximum displacement to the profile file. If using cyclic symmetry analysis, this command will also write harmonic indices to the profile file.

For modal analysis, this command does not support the following mode-extraction methods (**MODOPT (p. 1140)**): unsymmetric matrix (UNSYM), the damped system (DAMP), or the QR-damped system (QRDAMP).

To write the normalized (instead of non-normalized) mode shapes from a modal analysis to the file:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use *Load* = MODE.
- Verify that the mode shapes are normalized to the mass matrix (**MODOPT (p. 1140)**,,,,,,OFF), the default behavior.
- Verify that the scale factor is set to 1.0 (**SET (p. 1724)**,,,1.0), the default value.

The nodes and underlying elements must be selected in order to be exported. See The Unidirectional Load-Transfer Method: Mechanical APDL to CFX for details.

For loads not specified directly via commands (such as **SF (p. 1733)** and **BF (p. 221)**), loads must first be read into the database (**SET (p. 1724)** or **LCASE (p. 1019)**).

Menu Paths

This command cannot be accessed from a menu.

EXPSOL, *LSTEP*, *SBSTEP*, *TIMFRQ*, *Elcalc*

Specifies the solution to be expanded for mode-superposition analyses or substructure analyses.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSTEP, SBSTEP

Expand the solution identified as load step *LSTEP* and substep *SBSTEP*.

TIMFRQ

As an alternative to *LSTEP* and *SBSTEP*, expand the solution at, or nearest to, the time value *TIMFRQ* (for **ANTYPE (p. 162)**,TRANS or **ANTYPE (p. 162)**,SUBSTR) or frequency value *TIMFRQ* (for **ANTYPE (p. 162)**,HARMIC). *LSTEP* and *SBSTEP* should be blank.

Elcalc

Element calculation key:

YES

Calculate element results, nodal loads, and reaction loads.

NO

Do not calculate these items.

Notes

Specifies the solution to be expanded from analyses that use the mode-superposition method (**ANTYPE (p. 162)**,HARMIC or TRANS) or substructuring (**ANTYPE (p. 162)**,SUBSTR). Use the **NUMEXP (p. 1293)** command to expand a group of solutions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The resulting results file will maintain the same load step, substep, and time (or frequency) values as the requested solution to be expanded.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Load Step

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Load Step

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq

---

EXTOPT, *Lab*, *Val1*, *Val2*, *Val3*, *Val4*

Controls options relating to the generation of volume elements from area elements.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Label identifying the control option. The meanings of *Val1*, *Val2*, and *Val3* will vary depending on *Lab*.

ON

Sets carryover of the material attributes, real constant attributes, and element coordinate system attributes of the pattern area elements to the generated volume elements. Sets the pattern area mesh to clear when volume generations are done. *Val1*, *Val2*, and *Val3* are ignored.

OFF

Removes all settings associated with this command. *Val1*, *Val2*, and *Val3* are ignored.

STAT

Shows all settings associated with this command. *Val1*, *Val2*, *Val3*, and *Val4* are ignored.

ATTR

Sets carryover of particular pattern area attributes (materials, real constants, and element coordinate systems) of the pattern area elements to the generated volume elements. (See 2.) *Val1* can be:

0

Sets volume elements to use current **MAT (p. 1111)** command settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Sets volume elements to use material attributes of the pattern area elements.

Val2 can be:

0

Sets volume elements to use current **REAL (p. 1579)** command settings.

1

Sets volume elements to use real constant attributes of the pattern area elements.

Val3 can be:

0

Sets volume elements to use current **ESYS (p. 685)** command settings.

1

Sets volume elements to use element coordinate system attributes of the pattern area elements.

Val4 can be:

0

Sets volume elements to use current **SECNUM (p. 1685)** command settings.

1

Sets volume elements to use section attributes of the pattern area elements.

ESIZE

Val1 sets the number of element divisions in the direction of volume generation or volume sweep. For **VDRAG (p. 2084)** and **VSWEEP (p. 2168)**, *Val1* is overridden by the **LESIZE (p. 1041)** command *NDIV* setting. *Val2* sets the spacing ratio (bias) in the direction of volume generation or volume sweep. If positive, *Val2* is the nominal ratio of last division size to first division size (if > 1.0, sizes increase, if < 1.0, sizes decrease). If negative, *Val2* is the nominal ratio of center division(s) size to end divisions size. Ratio defaults to 1.0 (uniform spacing). *Val3* and *Val4* are ignored.

ACLEAR

Sets clearing of pattern area mesh. (See 3.) *Val1* can be:

0

Sets pattern area to remain meshed when volume generation is done.

1

Sets pattern area mesh to clear when volume generation is done. *Val2*, *Val3* , and *Val4* are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VSWE

Indicates that volume sweeping options will be set using *Val1* and *Val2*. Settings specified with **EXTOPT (p. 722)**,VSWE will be used the next time the **VSWEEP (p. 2168)** command is invoked. If *Lab* = VSWE, *Val1* becomes a label. *Val1* can be:

AUTO

Indicates whether you will be prompted for the source and target used by **VSWEEP (p. 2168)** or if VSWE should automatically determine the source and target. If *Val1* = AUTO, *Val2* is ON by default. VSWE will automatically determine the source and target for **VSWEEP (p. 2168)**. You will be allowed to pick more than one volume for sweeping. When *Val2* = OFF, the application prompts you for the source and target for **VSWEEP (p. 2168)**. You will only be allowed to pick one volume for sweeping.

TETS

Indicates whether **VSWEEP (p. 2168)** will tet mesh non-sweepable volumes or leave them unmeshed. If *Val1* = TETS, *Val2* is OFF by default. Non-sweepable volumes will be left unmeshed. When *Val2* = ON, the non-sweepable volumes will be tet meshed if the assigned element type supports tet shaped elements.

Val3 is ignored for *Lab* = VSWE.

Val1*, *Val2*, *Val3*, *Val4

Additional input values as described under each option for *Lab*.

Notes

1. **EXTOPT (p. 722)** controls options relating to the generation of volume elements from pattern area elements using the **VEXT (p. 2091)**, **VROTAT (p. 2155)**, **VOFFST (p. 2140)**, **VDRAG (p. 2084)**, and **VSWEEP (p. 2168)** commands. (When using **VSWEEP (p. 2168)**, the pattern area is referred to as the source area.)

2. Enables carryover of the attributes of the pattern area elements to the generated volume elements when you are using **VEXT (p. 2091)**, **VROTAT (p. 2155)**, **VOFFST (p. 2140)**, or **VDRAG (p. 2084)**. (When using **VSWEEP (p. 2168)**, since the volume already exists, use the **VATT (p. 2075)** command to assign attributes before sweeping.)

3. When you are using **VEXT (p. 2091)**, **VROTAT (p. 2155)**, **VOFFST (p. 2140)**, or **VDRAG (p. 2084)**, enables clearing of the pattern area mesh when volume generations are done. (When you are using **VSWEEP (p. 2168)**, if selected, the area meshes on the pattern (source), target, and/or side areas clear when volume sweeping is done.)

4. Neither **EXTOPT (p. 722)**,VSWE,AUTO nor **EXTOPT (p. 722)**,VSWE,TETS will be affected by **EXTOPT (p. 722)**,ON or **EXTOPT (p. 722)**, OFF.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volume Sweep>Sweep Opts

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Operate>Extrude>Elem Ext Opts

---

EXTREM, *NVAR1*, *NVAR2*, *NINC*

Lists the extreme values for variables.

POST26 (p. 58): Listing (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR1, NVAR2, NINC

List extremes for variables *NVAR1* through *NVAR2* in steps of *NINC*. Variable range defaults to its maximum. *NINC* defaults to 1.

Notes

Lists the extreme values (and the corresponding times) for stored and calculated variables. Extremes for stored variables are automatically listed as they are stored. Only the real part of a complex number is used. Extreme values may also be assigned to parameters (***GET (p. 797)**).

Menu Paths

Main Menu>TimeHist Postpro>List Extremes

EXUNIT, *Ldtype*, *Load*, *Untype*, *Name*

Specifies the interface data unit labels to be written to the profile file from Mechanical APDL to Ansys CFX transfer.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ldtype

Load type:

SURF

Surface load.

VOLU

Volumetric load.

Load

Surface loads:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DISP

Displacement in a static analysis. Mode shape in a modal analysis.

TIME

Time. The unit for frequency is the inverse of the unit for time.

MASS

Mass.

TEMP

Temperature.

HFLU

Heat flux.

Volumetric loads:

DISP

Displacement.

FORC

Force

HGEN

Heat generation

Untype

Unit type:

COMM

Predefined unit

USER

User-specified unit

Name

Commonly used predefined unit name or user-specified unit name.

SI

International System of units (meter-kilogram-second) (default)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FT

English System of units (feet-pound-second)

In the SI system, surface loads are in units of m for DISP, degrees K for TEMP, and W/m^2 for HFLU; volumetric loads are in units of m for DISP, N/m^3 for FORC, and W/m^3 for HGEN.

In the English system, surface loads are in units of ft for DISP, degrees F for TEMP, and $BTU/sec\text{-}ft^2$ for HFLU; volumetric loads are in units of ft for DISP, pdl/ft^3 for FORC, and $BTU/sec\text{-}ft^3$ for HGEN. A pdl is a poundal, and 32.174 pdl = 1 lbf.

Notes

This command only specifies which unit labels are to be written to the file when the **EXPROFILE (p. 719)** is issued. It does not perform unit conversions.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

F Commands

F, *Node*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*, –, –, *MESHFLAG*

Defines force loads at nodes.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Node

Node at which force is to be specified. If ALL, *NEND* and *NINC* are ignored and forces are applied to all selected nodes (**NSEL (p. 1266)**). If *Node* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *Node*.

Lab

Valid force labels are:

Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments).

Thermal labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow).

Fluid label: FLOW (fluid flow).

Electric labels: AMPS (current flow), CHRG (electric charge).

Magnetic labels: FLUX (magnetic flux); CSGZ (magnetic current segment).

Diffusion label: RATE (diffusion flow rate).

Viscous-thermal acoustics labels: FX, FY, FZ (volumetric force density).

For structural analyses, DVOL (fluid mass flow rate) is also a valid label. See Notes (p. 730) for more information.

VALUE

Force value or table name reference for specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), for example, **F (p. 729)**, *Node*,HEAT,%*tabname*%). To define a table, issue ***DIM (p. 530)**.

VALUE2

Second force value (if any). If the analysis type and the force allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NEND, NINC

Specifies the same values of force at the nodes ranging from *Node* to *NEND* (defaults to *Node*), in steps of *NINC* (defaults to 1).

–

Reserved.

–

Reserved.

MESHFLAG

Specifies how to apply nodal force on the mesh. Valid in a nonlinear adaptivity analysis when *Lab* = FX / FY / FZ and *Node* is not a component name. Not valid when *Lab* = ALL.

0 – Nodal-force loading occurs on the current mesh (default).

1 – Nodal-force loading occurs on the initial mesh for nonlinear adaptivity. (*NEND* and *NINC* are not valid.)

Notes

The available force loads per node correspond to the degrees of freedom listed under **Degrees of Freedom** in the input table for each element type in the *Element Reference*. If both a force and a constrained degree of freedom (**D (p. 483)**) are specified at the same node, the constraint takes precedence. Forces are defined in the nodal coordinate system. The positive directions of structural forces and moments are along and about the positive nodal axis directions. The node and the degree-of-freedom label corresponding to the force must be selected (**NSEL (p. 1266)**, **DOFSEL (p. 569)**).

Fluid flow (FLOW) is positive when flow is out of the nodes, and negative when flow is into the nodes.

For hydrostatic fluid elements (HSFLD241 and HSFLD242), DVOL is used to specify fluid mass flow rate (with units of mass/time) at the pressure node. This allows fluid to be added or taken out of the fluid elements sharing the pressure node. A fluid density must also be specified (**MP (p. 1160)** or **TB (p. 1899)**) to apply a volume change corresponding to the prescribed fluid mass flow rate.

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following labels: Fluid (FLOW), Electric (AMPS), Structural force (FX, FY, FZ, MX, MY, MZ), Thermal (HEAT, HBOT, HE2, HE3, . . ., HTOP), Diffusion (RATE). Tabular boundary conditions are valid only in static (**ANTYPE (p. 162)**,STATIC), full transient (**ANTYPE (p. 162)**,TRANS), full harmonic (**ANTYPE (p. 162)**, HARMIC), modal superposition harmonic and modal superposition transient analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharge>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCurrent>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>ImprCurr>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Node Components

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>NodePSD>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flow>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharge>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCurrent>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>ImprCurr>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Node Components

Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>NodePSD>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Nodes

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flow>On Nodes

---

/FACET, *Lab*

Specifies the facet representation used to form solid model displays.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid labels:

FINE

Use finer tessellation to increase the number of facets for the display. Provides the best representation (but decreases speed of operation).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NORML

Use the basic number of facets for the display (default).

COAR

Use a limited number of facets for the display. This option will increase the speed of the operations, but may produce poor representations for some imported models.

WIRE

Display model with a wireframe representation (fast, but surfaces will not be shown).

Command Default

Basic number of facets.

Notes

Specifies the facet (or polygon) representation used to form solid model displays. Used only with the **APLOT (p. 169)**, **ASUM (p. 203)**, **VPLOT (p. 2147)**, and **VSUM (p. 2167)** commands.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Solid Model Facets

FC, *MAT*, *Lab1*, *Lab2*, *DATA1*, *DATA2*, *DATA3*, *DATA4*, *DATA5*, *DATA6*

Provides failure criteria information and activates a data table to input temperature-dependent stress and strain limits.

PREP7 (p. 22): Materials (p. 24)
POST1 (p. 51): Failure Criteria (p. 58)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Material reference number. You can define failure criteria for up to 250 different materials.

Lab1

Type of data.

TEMP

Temperatures. Each of the materials you define can have a different set of temperatures to define the failure criteria.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EPEL

Strains.

S

Stresses.

Lab2

Specific criteria. Not used if *Lab1* = TEMP.

XTEN

Allowable tensile stress or strain in the x-direction. (Must be positive.)

XCMP

Allowable compressive stress or strain in the x-direction. (Defaults to negative of XTEN.)

YTEN

Allowable tensile stress or strain in the y-direction. (Must be positive.)

YCMP

Allowable compressive stress or strain in the y-direction. (Defaults to negative of YTEN.)

ZTEN

Allowable tensile stress or strain in the z-direction. (Must be positive.)

ZCMP

Allowable compressive stress or strain in the z-direction. (Defaults to negative of ZTEN.)

XY

Allowable XY stress or shear strain. (Must be positive.)

YZ

Allowable YZ stress or shear strain. (Must be positive.)

XZ

Allowable XZ stress or shear strain. (Must be positive.)

XYCP

XY coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0. [1]

YZCP

YZ coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0. [1]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

XZCP

XZ coupling coefficient (Used only if *Lab1* = S). Defaults to -1.0. [1]

XZIT

XZ tensile inclination parameter for Puck failure index (default = 0.0)

XZIC

XZ compressive inclination parameter for Puck failure index (default = 0.0)

YZIT

YZ tensile inclination parameter for Puck failure index (default = 0.0)

YZIC

YZ compressive inclination parameter for Puck failure index (default = 0.0)

G1G2

Fracture toughness ratio between GI (mode I) and GII (mode II)

ETAL

Longitudinal friction coefficient

ETAT

Transverse friction coefficient

APL0

Fracture angle under pure transverse compression (default to 53) [2]

1. Entering a blank or a zero for XYCP, YZCP, or XZCP triggers the default value of -1.0. To specify an effective zero, use a small, nonzero value (such as 1E-14) instead. For more information, see Command and Argument Defaults (p. 7).

2. Entering a blank or a zero ALP0 triggers the default value of 53. To specify an effective zero, use a small, nonzero value (such as 1E-14) instead. For more information, see Command and Argument Defaults (p. 7).

DATA1*, *DATA2*, *DATA3*, . . . , *DATA6

Description of *DATA1* through *DATA6*.

T1, T2, T3, T4, T5, T6

Temperature at which limit data is input. Used only when *Lab1* = TEMP.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

V1, V2, V3, V4, V5, V6

Value of limit stress or strain at temperature T1 through T6. Used only when *Lab1* = S or EPEL.

Notes

The data table can be input in either PREP7 or POST1. This table is used only in POST1. When you postprocess failure criteria results defined via the **FC (p. 732)** command (**PLESOL (p. 1401)**, **PRESOL (p. 1481)**, **PLNSOL (p. 1425)**, **PRNSOL (p. 1505)**, **PRRSOL (p. 1518)**, etc.), the active coordinate system must be the coordinate system of the material being analyzed. You do this using **RSYS (p. 1639)**, SOLU. For layered applications, you also use the **LAYER (p. 1015)** command. See the specific element documentation in the *Element Reference* for information about defining your coordinate system for layers.

Some plotting and printing functions will not support Failure Criteria for your PowerGraphics displays. This could result in minor changes to other data when Failure Criteria are applied. See the appropriate plot or print command documentation for more information.

Menu Paths

Main Menu>General Postproc>Failure Criteria>Add/Edit

Main Menu>General Postproc>Failure Criteria>Temp Variation

Main Menu>Preprocessor>Material Props>Failure Criteria>Add/Edit

Main Menu>Preprocessor>Material Props>Failure Criteria>Temp Variation

---

FCCHECK

Checks both the strain and stress input criteria for all materials.

PREP7 (p. 22): Materials (p. 24)
POST1 (p. 51): Failure Criteria (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Issue the **FCCHECK (p. 735)** command to check the completeness of the input during the input phase.

Menu Paths

Main Menu>General Postproc>Failure Criteria>Criteria Check

Main Menu>Preprocessor>Material Props>Failure Criteria>Criteria Check

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FCDELE, *MAT*

Deletes previously defined failure criterion data for the given material.

PREP7 (p. 22): Materials (p. 24)
POST1 (p. 51): Failure Criteria (p. 58)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Material number. Deletes all **FC (p. 732)** command input for *this* material.

A value of ALL deletes all **FC (p. 732)** command input for *all* materials.

Notes

This command is also valid in POST1.

Menu Paths

Main Menu>General Postproc>Failure Criteria>Delete

Main Menu>Preprocessor>Material Props>Failure Criteria>Delete

/FCLEAN

Deletes all local files in all processors in a distributed parallel processing run.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Deletes all local files (`.rst`, `.esav`, `.emat`, `.mode`, `.mlv`, `.seld`, `.dsub`, `.ist`, `.full`, `.rdsp`, `.rfrq`, `.rnnn`, `.resf`, `.stat`, `.modesym`, `.osave`, `.erot`, `.log`) in all processors in a distributed parallel processing run.

Notes

Issue **/FCLEAN (p. 736)** to delete all local files having the current *Jobname* (**/FILNAME (p. 757)**) and save disk space in a distributed parallel processing run. Like other file deletion commands, deletion happens immediately upon issuing this command. Different than other file deletion commands, it enables the convenience of deleting all *Jobname*`.*` local files without having to issue separate commands specifying each file type.

All `.log` files except the master (*Jobname*`0.log`) are deleted.

Caution:

Because **/FCLEAN (p. 736)** deletes all local files, it should only be issued if you are sure that none of those files are needed in any downstream analyses. Deleting files that are necessary

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for subsequent substeps, load steps, commands, or analyses will prevent continuation of the run. For example, since the local files are combined into global files when you issue **FINISH (p. 758)** in the solution processor, issuing **/FCLEAN (p. 736)** before **FINISH (p. 758)** in **/SOLU (p. 1821)** will result in a program crash.

Menu Paths

This command cannot be accessed from a menu.

FCLIST, *MAT*, - -, *TEMP*

To list what the failure criteria is that you have input.

PREP7 (p. 22): Materials (p. 24)
POST1 (p. 51): Failure Criteria (p. 58)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Material number (defaults to ALL for all materials).

- -

Unused field.

TEMP

Temperature to be evaluated at (defaults to TUNIF).

Notes

This command allows you to see what you have already input for failure criteria using the FC commands.

Menu Paths

Main Menu>General Postproc>Failure Criteria>List

Main Menu>Preprocessor>Material Props>Failure Criteria>List

/FCOMP, *Ident*, *LEVEL*

Specifies file-compression options.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ident

Mechanical APDL file identifier. There is no default. Valid labels are:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RST

Results file.

DB

Database file.

RNNN

Restart file.

OSAV

File created during a nonlinear analysis that contains a copy of ESAV file from the last converged substep.

LEVEL

Compression level:

SPARSE

Use a sparsification scheme for file compression (default).

0

No file compression occurs.

n

A zlib-based file compression occurs using level number *n*, which ranges from 1 to 5.

Command Default

Files listed under `Ident` are compressed by default using the sparsification scheme.

Notes

Specifies file compression options for results files (`.rst`, `.rstp`, `.rth`, and `.rmg` files), database files (`.db` and `.rdb`), certain restart files (`.Rnnn`), and the `.osav` file created during a nonlinear analysis. (See Program-Generated Files in the *Basic Analysis Guide* for file descriptions.) Records are compressed as they are written and uncompressed as they are read (for example, by the **SET (p. 1724)** command or the **RESUME (p. 1601)** command).

For results files compressed using the sparsification scheme (*LEVEL* = SPARSE, which is the default), use the ***XPL (p. 2196)** command to uncompress the file. For third party tools that need to read the results file, use the method described in Accessing Mechanical APDL Binary Files, which automatically decodes any SPARSE compressed records in the results file.

See File Compression in the *Basic Analysis Guide* for more details.

This command is valid only at the Begin Level.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

FCUM,*Oper*,*RFACT*,*IFACT*

Specifies that force loads are to be accumulated.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Accumulation key:

REPL

Subsequent values replace the previous values (default).

ADD

Subsequent values are added to the previous values.

IGNO

Subsequent values are ignored.

RFACT

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

Command Default

Replace previous values.

Notes

Allows repeated force load (force, heat flow, etc.) values to be replaced, added, or ignored. Operations apply to the selected nodes [NSEL]. and the force labels corresponding to the selected force labels (**DOFSEL (p. 569)**). The operations occur when the next force specifications are defined. For example, issuing the command **F (p. 729)**,1,FX,250 after a previous **F (p. 729)**,1,FX,200 causes the current value of the force on node 1 in the x-direction to be 450 with the add operation, 250 with the replace operation, or 200 with the ignore operation. Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a force of 700. Scale factors are applied even if no previous values exist. Issue **FCUM (p. 739)**,STAT to show the current

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note:

FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

FCUM (p. 739) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Forces

Main Menu>Solution>Define Loads>Settings>Replace vs Add>Forces

---

FCTYP, *Oper*, *Lab*

Activates or removes failure-criteria types for postprocessing.

POST1 (p. 51): Failure Criteria (p. 58)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Operation key:

ADD

Activate failure-criteria types. This option is the default behavior.

DELE

Remove failure-criteria types.

Lab

Valid failure-criteria labels. If ALL, select all available (including user-defined) failure criteria.

EMAX

Maximum strain criterion (default)

SMAX

Maximum stress criterion (default)

TWSI

Tsai-Wu strength index (default)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TWSR

Inverse of Tsai-Wu strength ratio index (default)

HFIB

Hashin fiber failure criterion

HMAT

Hashin matrix failure criterion

PFIB

Puck fiber failure criterion

PMAT

Puck inter-fiber (matrix) failure criterion

L3FB

LaRc03 fiber failure criterion

L3MT

LaRc03 matrix failure criterion

L4FB

LaRc04 fiber failure criterion

L4MT

LaRc04 matrix failure criterion

USR1 through USR9

User-defined failure criteria

Command Default

FCTYP (p. 740), ADD, EMAX, SMAX, TWSI, TWSR

Notes

The **FCTYP (p. 740)** command modifies the list of active failure criteria.

By default, active failure criteria include EMAX, SMAX, TWSI, and TWSR.

The command affects any subsequent postprocessing listing and plotting commands (such as **PRESOL (p. 1481)**, **PRNSOL (p. 1505)**, **PLESOL (p. 1401)**, **PLNSOL (p. 1425)**, and **ETABLE (p. 687)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A single **FCTYP (p. 740)** command allows up to six failure-criteria labels. If needed, reissue the command to activate or remove additional failure-criteria types.

Menu Paths

This command cannot be accessed from a menu.

FDELE, *NODE*, *Lab*, *NEND*, *NINC*, *Lkey*

Deletes force loads on nodes.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node for which force is to be deleted. If ALL, *NEND* and *NINC* are ignored and forces are deleted on all selected nodes (**NSEL (p. 1266)**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab

If ALL, use all appropriate labels. Valid force labels are:

Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments).

Thermal labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow).

Fluid label: FLOW (fluid flow).

Electric labels: AMPS (current flow), CHRG (electric charge).

Magnetic labels: FLUX (magnetic flux); CSGZ (magnetic current segment).

Diffusion label: RATE (diffusion flow rate).

NEND, NINC

Delete forces from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1).

Lkey

Lock key:

(blank)

The DOF is not locked (default).

FIXED

Displacement on the specified degrees of freedom (*Lab*) is locked. The program prescribes the degree of freedom to the "current" relative displacement value in addition to deleting the force. If a displacement constraint (for example, **D (p. 483)** command) is applied in conjunction with this option, the actual applied displacement will be ramped during the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

next load step. The displacement is ramped from the current value to the newly defined value. This option is only valid for the following labels: FX, FY, FZ, MX, MY, MZ. This option is intended primarily for use in the Ansys Workbench interface to apply an increment length adjustment (bolt pretension loading).

Notes

The node and the degree of freedom label corresponding to the force must be selected (**NSEL (p. 1266)**, **DOFSEL (p. 569)**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On All Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharge>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCurrent>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>ImprCurr>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Flow>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Node Components

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>NodePSD>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flow>On Nodes

Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On All Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharge>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCurrent>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>ImprCurr>On Nodes

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Flow>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Node Components

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>NodePSD>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Nodes

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flow>On Nodes

---

/FDELE, *Ident*, *Stat*

Deletes a binary file after it is used.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ident

Mechanical APDL file name identifier. Valid identifiers are: EMAT, ESAV, FULL, SUB, MODE, DSUB, USUB, OSAV, and SELD. See the *Basic Analysis Guide* for file descriptions.

Stat

Keep or delete key:

KEEP

Keep this file.

DELE

Delete (or do not write, if not necessary) this file.

Command Default

Keep all files.

Notes

Deletes as soon as possible (or prevents writing) a binary file created by Mechanical APDL to save space.

Caution:

Deleting files that are necessary for the next substep, load step, or analysis will prevent continuation of the run.

This command is valid only at the Begin level.

Menu Paths

Utility Menu>File>File Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FEBODY

Specifies "Body loads on elements" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Body Loads

FECONS

Specifies "Constraints on nodes" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>DOF Constraints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FEFOR

Specifies "Forces on nodes" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Forces

FESURF

Specifies "Surface loads on elements" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Surface Loads

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*FFT, *Type*, *InputData*, *OutputData*, *DIM1*, *DIM2*, *ResultFormat*

Computes the fast Fourier transformation of a specified matrix or vector.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Type

Type of FFT transformation:

FORW --

Forward FFT computation (default).

BACK --

Backward FFT computation.

InputData

Name of matrix or vector for which the FFT will be computed. This can be a dense matrix (created by the ***DMAT (p. 551)** command) or a vector (created by the ***VEC (p. 2085)** command). Data can be real or complex values. There is no default value for this argument.

OutputData

Name of matrix or vector where the FFT results will be stored. The type of this argument must be consistent with *InputData* (see table below). There is no default value for this argument.

FFT Domain	Forward		Backward	
	Input Data	**Output Data**	**Input Data**	**Output Data**
Real	Real	Complex	Complex	Real
Complex	Complex	Complex	Complex	Complex

DIM1

The number of terms to consider for a vector, or the number of rows for a matrix. Defaults to the whole input vector or all the rows of the matrix.

DIM2

The number of columns to consider for a matrix. Defaults to all the columns of the matrix. (Valid only for matrices.)

ResultFormat

Specifies the result format:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FULL --

Returns the full result. That is, the result matches the dimension specified on this command (*DIM1*, *DIM2*).

PART --

Returns partial results. For real input data, there is a symmetry in the results of the Fourier transform as some coefficients are conjugated. The partial format uses this symmetry to optimize the storage of the results. (Valid only for real data.)

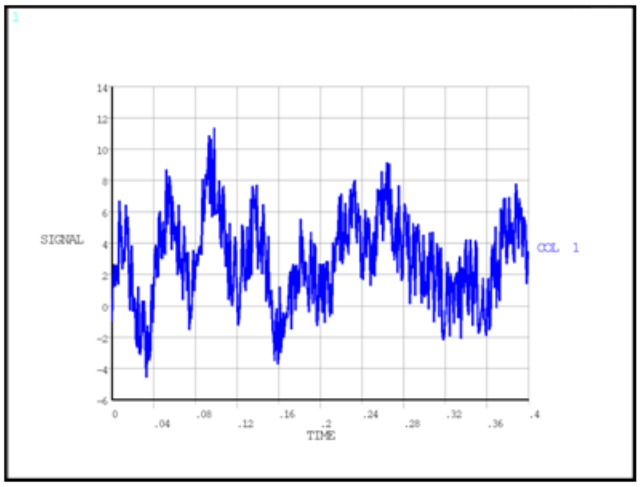
Notes

In the example that follows, the fast Fourier transformation is used to filter frequencies from a noisy input signal.

Example Usage

The fast Fourier transformation can be used to create a filter to remove noise from a noisy input signal.

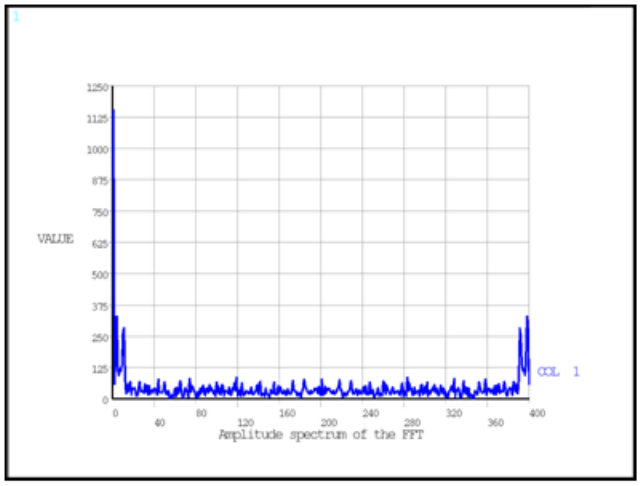
Figure 4: Input Signal



The input signal is decomposed in the Fourier space using the ***FFT (p. 747)**,*FORW* command so that the frequencies that compose the signal can be observed.

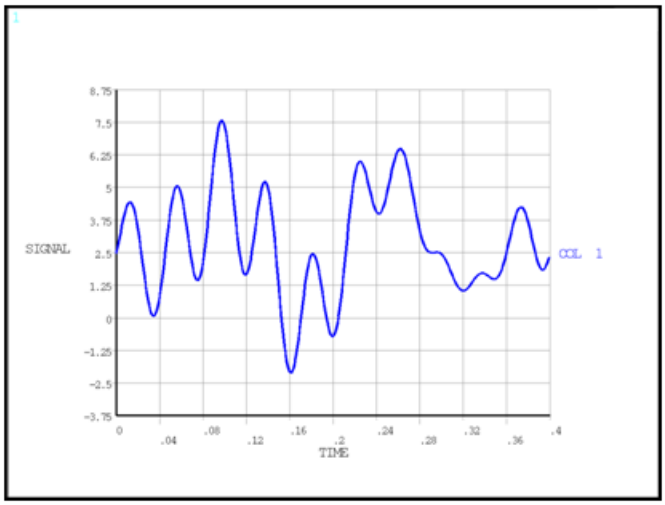
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5: Amplitude Spectrum of the FFT



The filter applies a threshold on the Fourier coefficients. Fourier coefficients of small amplitude are the result of the noise in the input signal. Only coefficients of a large amplitude are significant. The inversion of the ***FFT (p. 747)** command (***FFT (p. 747)**,*BACK* command) rebuilds the signal from the remaining coefficients.

Figure 6: Signal Obtained with the Filter



Example input demonstrating ***FFT (p. 747)** usage is provided below:

```
/com,************************************************
/com,*  Usage example of the *FFT APDLMath command.  *
/com,*  Use FFT to filter a noisy sinusoidal signal. *
/com,************************************************
/verify
/nopr

/post26

/gopr
/com, *** Definition of the parameters for the signal

pi              = acos(-1)
periode1        = 2*pi/40.0
phase1          = 4.0
amplitude1      = 2.5
```

```
periode2       = 2*pi/150.0
phase2         = 0.0
amplitude2     = 2.0
periode3       = 2*pi/140.0
phase3         = 0.0
amplitude3     = 1.5
amplitudenoise = 6.0
n              = 400       !number of points in the sampling input signal
tbegin         = 0.0       !start time for the signal
dt             = 0.001     !time step
tend           = tbegin + (n-1)*dt


/com, *** Definition of the parameters

threshold = 150.0
FFTmethod = 1 !0 = FFT giving partial results , 1 = FFT giving full results


/com, *** Create a signal from two sinus and add noise

*dim,signal,array,n
*dim,t,array,n
*dim,noise,array,n
*vfill,noise,rand

omega1 = (2.0*pi) / periode1
omega2 = (2.0*pi) / periode2
omega3 = (2.0*pi) / periode3

*do,ii,1,n
  noisec     =  amplitudenoise * noise(ii)
  tc         =  tbegin + (ii-1)*dt
  t(ii)      =  tc
 signal(ii) = amplitude1*sin(omega1*tc+phase1) + amplitude2*sin(omega2*tc+phase2) + amplitude3*sin(omega3*tc+phase3
*enddo


/com, *** Display the input signal

*dim,signaldisp,table,n
*dim,tdisp,table,n

*do,ii,1,n
   tdisp(ii)=t(ii)
   signaldisp(ii)=signal(ii)
*enddo

/show,png
/AXLAB,X,TIME
/AXLAB,Y,SIGNAL
/color,curve,BLUE
/color,grbak,WHIT
/color,wbak,WHIT
/color,axes,BLAC
/color,axlab,BLAC
/color,axnum,blac
/GROPT,FILL,OFF
*vplot,tdisp,signaldisp


/com, *** Copy signal in an APDLMath vector

*vec,sigvec,d,alloc,n,1
*do,ii,1,n
 sigvec(ii) = signal(ii)
*enddo


/com, *** Perform a Fast Fourier Transform
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
*vec,VFFT,Z,ALLOC,n,1
*if,FFTmethod,eq,0,then
  *FFT,FORW,sigvec,VFFT,,,PART   !partial FFT (use symmetry properties)
*else
  *FFT,FORW,sigvec,VFFT,,,FULL   !full FFT
*endif


/com, *** Determine the real part and the imaginary part of the Fourier's coefficients

*vec,VFFTI,Z,COPY,VFFT
*AXPY,,,,0.,-1.,VFFTI
*vec,VI,D,COPY,VFFTI
*vec,VR,D,COPY,VFFT


/com, *** Compute the modulus of the Fourier's coeff

*EXPORT,VR,APDL,fftr
*EXPORT,VI,APDL,ffti

sizeVR = VR_rowDim

*if,FFTmethod,eq,0,then
  sizeVR = sizeVR/2
*endif

*dim,ind,table,sizeVR
*dim,modfft,array,sizeVR

*do,ii,1,sizeVR
  ind(ii)   = ii
  modfft(ii)= sqrt( fftr(ii)*fftr(ii) + ffti(ii)*ffti(ii) )
*enddo


/com,*** Display the Amplitude spectrum of the FFT

*dim,modfftdisp,table,sizeVR
*do,ii,1,sizeVR
   modfftdisp(ii) = modfft(ii)
*enddo

/show,png
/AXLAB,X,Amplitude spectrum of the FFT
/AXLAB,Y,
*vplot,ind,modfftdisp
fini


/com, *** Threshold on Fourier coefficients

*print,VR
*do,ii,1,sizeVR
  module = modfft(ii)
  *if,module,lt,threshold,then
    VR(ii) = 0.0
    VI(ii) = 0.0
  *endif
*enddo


/com, *** Invert Fourier Transform

*vec,VIFT,Z,COPY,VR
*AXPY,0.0,1.0,VI,1.0,0.0,VIFT
*vec,VFILTER,D,ALLOC,VR_DIM

*FFT,BACK,VIFT,VFILTER
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/com, *** Display the filtered signal

*EXPORT,VFILTER,APDL,filterSignal
*dim,fsignal,table,sizeVR

*do,ii,1,sizeVR
   fsignal(ii) = filterSignal(ii)
*enddo

/show,png
/AXLAB,X,TIME
/AXLAB,Y,SIGNAL
*vplot,tdisp,fsignal
fini

/com, *** Free the APDLMath objects
*free,all
```

Menu Paths

This command cannot be accessed from a menu.

FILE,*Fname*,*Ext*,--

Specifies the data file where results are to be found.

POST1 (p. 51): Set Up (p. 51)
POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

If *Fname* is blank, the extension defaults to RST (for structural, fluid, or coupled-field analyses), to RTH (for thermal or electrical analyses), or to RMG (for magnetic analyses). For postprocessing contact results corresponding to the initial contact state in POST1, use the RCN extension. For postprocessing modal coordinates results in POST1, use the RDSP or RFRQ extension.

--

Unused field.

Command Default

Use the result file with the Jobname as *Fname* and with the extension corresponding to the analysis type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Specifies the Mechanical APDL data file where the results are to be found for postprocessing.

Issuing the **FILE (p. 752)** command with *Ext* = RSDP or in POST1 specifies the `.rsdp` or `.rfrq` file used by the **PRMC (p. 1499)** and **PLMC (p. 1421)** commands. (See POST1 example (p. 753) below.)

Example Usage

POST1

```
/POST1
FILE,,rdsp   !Choose file Jobname.rdsp from a previous MSUP transient solution
PRMC,2,1     !Plot Modal coordinates from rdsp file (loadstep and substep specified as arguments)
FILE,,rst    !Choose file Jobname.rst
SET,1,1      !Load results from chosen .rst file into database
PRNSOL,u,x   !Plot Nodal displacements (loadstep and substep specified using the SET command)
FINISH       !Exit POST1
```

POST26

Example: Mode-Superposition Transient Dynamic Analysis

Menu Paths

Main Menu>General Postproc>Data & File Opts

Main Menu>TimeHist Postpro>Settings>File

Utility Menu>File>List>Binary Files

Utility Menu>List>Files>Binary Files

FILEAUX2, *Fname*, *Ident*, --

Specifies the binary file to be dumped.

AUX2 (p. 61): Binary File Dump (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current `Jobname` if *Ident* is specified.

Ident

File name identifier. See the *Basic Analysis Guide* for file descriptions and identifiers. If not an identifier, the program uses *Ident* as the file name extension.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--

Unused field.

Notes

Specifies the binary file to be dumped with the **DUMP (p. 583)** command.

Menu Paths

Utility Menu>File>List>Binary Files

Utility Menu>List>Files>Binary Files

FILEAUX3, *Fname*, *Ext*, - -

Specifies the results file to be edited.

AUX3 (p. 61): Results Files (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current Jobname if *Ext* is specified.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Notes

Specifies the results file to be edited.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FILL,*NODE1*,*NODE2*,*NFILL*,*NSTRT*,*NINC*,*ITIME*,*INC*,*SPACE*

Generates a line of nodes between two existing nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2

Beginning and ending nodes for fill-in. *NODE1* defaults to next to last node specified, *NODE2* defaults to last node specified. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NFILL

Fill *NFILL* nodes between *NODE1* and *NODE2* (defaults to |*NODE2*-*NODE1*|-1). *NFILL* must be positive.

NSTRT

Node number assigned to first filled-in node (defaults to *NODE1* + *NINC*).

NINC

Add this increment to each of the remaining filled-in node numbers (may be positive or negative). Defaults to the integer result of (*NODE2*-*NODE1*)/(*NFILL* + 1), that is, linear interpolation. If the default evaluates to zero, or if zero is input, *NINC* is set to 1.

ITIME*, *INC

Do fill-in operation a total of *ITIMEs*, incrementing *NODE1*, *NODE2* and *NSTRT* by *INC* each time after the first. *ITIME* and *INC* both default to 1.

SPACE

Spacing ratio. Ratio of last division size to first division size. If > 1.0, divisions increase. If < 1.0, divisions decrease. Ratio defaults to 1.0 (uniform spacing).

Notes

Generates a line of nodes (in the active coordinate system) between two existing nodes. The two nodes may have been defined in any coordinate system. Nodal locations and rotation angles are determined by interpolation. Any number of nodes may be filled-in and any node number sequence may be assigned. See the **CSCIR (p. 436)** command when filling across the 180° singularity line in a non-Cartesian system.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Fill between Nds

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FILLDATA, *IR*, *LSTRT*, *LSTOP*, *LINC*, *VALUE*, *DVAL*

Fills a variable by a ramp function.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Define data table as variable *IR* (2 to *NV* (**NUMVAR (p. 1302)**)).

LSTRT

Start at location *LSTRT* (defaults to 1).

LSTOP

Stop at location *LSTOP* (defaults to maximum location as determined from data previously stored.

LINC

Fill every *LINC* location between *LSTRT* and *LSTOP* (defaults to 1).

VALUE

Value assigned to location *LSTRT*.

DVAL

Increment value of previous filled location by *DVAL* and assign sum to next location to be filled (may be positive or negative.)

Notes

Locations may be filled continuously or at regular intervals (*LINC*). Previously defined data at a location will be overwritten.

Menu Paths

Main Menu>TimeHist Postpro>Table Operations>Fill Data

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/FILNAME, *Fname*, *Key*

Changes the Jobname for the analysis.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

Name (32 characters maximum) to be used as the Jobname. Defaults to the initial Jobname as specified on the Mechanical APDL execution command, or to file if none specified.

Key

Specify whether to use the existing log, error, lock, page, and output files (.log, .err, .lock, .page and .out) or start new files.

0, OFF

Continue using current log, error, lock, page, and output files.

1, ON

Start new log, error, lock, page, and output files (old log and error files are closed and saved, but old lock, page, and output files are deleted). Existing log and error files are appended.

Notes

All subsequently created files will be named with this Jobname if *Key* = 0. Use *Key* = 1 to start new log, error, lock, page, and output files. The previous Jobname is typically defined on the Mechanical APDL program execution line. (See the *Operations Guide*.

This command is useful when different groups of files created throughout the run are to have different names. For example, the command may be used before each substructure pass to avoid overwriting files or having to rename each file individually.

This command is valid only at the Begin level.

Menu Paths

Utility Menu>File>Change Jobname

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FINISH

Exits normally from a processor.

SESSION (p. 11): Processor Entry (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command exits any of the Mechanical APDL processors.

When exiting the Mechanical APDL processors, data remains intact in the database, but the database is not automatically written to a file. (Issue **SAVE (p. 1645)** to write the database to a file.)

If exiting POST1 or POST26:

- POST1: Data in the database remains intact (including the POST1 element table data, the path table data, the fatigue table data, and the load case pointers).
- POST26: Data in the database remains intact, except that POST26 variables are erased and specification commands (such as **FILE (p. 752)**, **PRTIME (p. 1522)**, and **NPRINT (p. 1255)**) are reset. Issue **/QUIT (p. 1563)** to exit the processor and bypass these exceptions.

See **/QUIT (p. 1563)** for an alternate processor exit command.

This command is valid in any processor. This command is not valid at the Begin level.

Menu Paths

Main Menu>Finish

FITEM, *NFIELD*, *ITEM*, *ITEMY*, *ITEMZ*

Identifies items chosen by a picking operation (GUI).

DATABASE (p. 13): Picking (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFIELD

Field number on the command which uses the picking data. (Count the command name as a field, so that a 2 indicates the first command argument, 3 the second command argument, etc.) The corresponding field on the command will have a P51X label.

ITEM

Entity number of the entity picked. Negative entity numbers are used to indicate a range of entities. If the item picked is a coordinate location, then this field represents the X-coordinate. See also the **FLST (p. 766)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ITEMY*, *ITEMZ

Y and Z coordinates of a picked coordinate location. *ITEM* represents the X coordinate. See also the **FLST (p. 766)** command.

Notes

This is a command generated by the GUI and will appear in the log file (`Jobname.log`) if graphical picking is used. This command is *not* intended to be typed in directly in a session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

On the log file, a set of **FITEM (p. 758)** commands is preceded by one **FLST (p. 766)** command which defines the picking specifications for that pick operation. The data listed in the **FITEM (p. 758)** commands are used by the first subsequent command containing a P51X label in one of its fields.

Caution:

For a given entity type, a list containing an *ITEM* that is larger than the maximum defined entity, could deplete the system memory and produce unpredictable results.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

FJ, *ELEM*, *LABEL*, *VALUE*

Specify forces or moments on the components of the relative motion of a joint element.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element number or ALL to specify all joint elements.

LABEL

Valid labels:

FX

Force in local x direction.

FY

Force in local y direction.

FZ

Force in local z direction.

MX

Moment about local x axis.

MY

Moment about local y axis.

MZ

Moment about local z axis.

VALUE

Value of the label.

Notes

Valid for MPC184 (joint options in KEYOPT(1)).

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

See **FJDELE (p. 760)** for information on deleting forces and moments.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Joint Elems

Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Joint Elems

---

FJDELE, *ELEM*, *LAB*

Deletes forces (or moments) on the components of the relative motion of a joint element.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element number, or ALL. (leaving this blank defaults to ALL)

LAB

Valid labels are:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FX

Force in local x direction.

FY

Force in local y direction.

FZ

Force in local z direction.

MX

Moment about local x axis.

MY

Moment about local y axis.

MZ

Moment about local z axis.

ALL, or (blank)

Delete all valid forces or moments.

Notes

Valid for MPC184 (joint options in KEYOPT(1)).

See **FJ (p. 759)** for information on specifying forces (or moments).

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On Joint Elems

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Joint Elems

Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On Joint Elems

Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Joint Elems

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FJLIST, *Elem*

Lists forces and moments applied on joint elements.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Elem

Element number or ALL (or blank). Lists joint element forces and moments on the specified element(s).

Notes

Valid for MPC184 joint elements. See **FJ (p. 759)** for information on specifying forces and moments.

Menu Paths

Utility Menu>List>Loads>Joint Element Forces>On Picked Element

FK, *KPOI*, *Lab*, *VALUE*, *VALUE2*

Defines force loads at keypoints.

SOLUTION (p. 38): Solid Forces (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

Keypoint at which force is to be specified. If ALL, apply to all selected keypoints (**KSEL (p. 995)**). If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab

Valid force labels are:

Structural labels: FX, FY, or FZ (forces); MX, MY, or MZ (moments).

Thermal labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow).

Fluid label: FLOW (fluid flow).

Electric labels: AMPS (current flow), CHRG (electric charge).

Magnetic labels: FLUX (magnetic flux); CSGZ (magnetic current segment).

Diffusion label: RATE (diffusion flow rate).

VALUE

Force value or table name reference for specifying tabular boundary conditions. To specify a table, enclose the table name in percent signs (%), for example, **FK (p. 762)**, *KPOI*, HEAT,%*tabname*%). Use the ***DIM (p. 530)** command to define a table.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VALUE2

Second force value (if any). If the analysis type and the force allow a complex input, *VALUE* (above) is the real component and *VALUE2* is the imaginary component.

Notes

Forces may be transferred from keypoints to nodes with the **FTRAN (p. 783)** or **SBCTRAN (p. 1647)** commands. See the **F (p. 729)** command for a description of force loads.

Tabular boundary conditions (*VALUE* = %*tabname*%) are available only for the following labels: Fluid (FLOW), Electric (AMPS), Structural force (FX, FY, FZ, MX, MY, MZ), and Thermal (HEAT, HBOT, HE2, HE3, . . ., HTOP).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCharge>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppCurrent>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>ImprCurr>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>NodePSD>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flow>On Keypoints

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCharge>On Keypoints

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppCurrent>On Keypoints

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>ImprCurr>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppCurrSeg>On Keypoints

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMagFlux>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>MultiPtNod>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>NodePSD>On Keypoints

Main Menu>Solution>Define Loads>Apply>Structural>Spectrum>SinglPtFor>On Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flow>On Keypoints

FKDELE, *KPOI*, *Lab*

Deletes force loads at a keypoint.

SOLUTION (p. 38): Solid Forces (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

Keypoint at which force is to be deleted. If ALL, delete forces at all selected keypoints (**KSEL (p. 995)**). If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab

Valid force label. If ALL, use all appropriate labels. See the **FDELE (p. 742)** command for labels.

Notes

Deletes force loads (and all corresponding finite element loads) at a keypoint. See the **FDELE (p. 742)** command for details.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Forces>On All KPs

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCharge>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppCurrent>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>ImprCurr>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Flow>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Force/Moment>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>NodePSD>On Keypoints

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flow>On Keypoints
Main Menu>Solution>Define Loads>Delete>All Load Data>All Forces>On All KPs
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCharge>On Keypoints
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppCurrent>On Keypoints
Main Menu>Solution>Define Loads>Delete>Electric>Excitation>ImprCurr>On Keypoints
Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Flow>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppCurrSeg>On Keypoints
Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMagFlux>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Force/Moment>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>MultiPtNod>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>NodePSD>On Keypoints
Main Menu>Solution>Define Loads>Delete>Structural>Spectrum>SinglPtFor>On Keypoints
Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flow>On Keypoints

---

FKLIST, *KPOI*, *Lab*

Lists the forces at keypoints.

SOLUTION (p. 38): Solid Forces (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KPOI

List forces at this keypoint. If ALL (default), list for all selected keypoints (**KSEL (p. 995)**). If *KPOI* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *KPOI*.

Lab

Force label to be listed (defaults to ALL). See the **DOFSEL (p. 569)** command for labels.

Notes

Listing applies to the selected keypoints (**KSEL (p. 995)**) and the selected force labels (**DOFSEL (p. 569)**).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Forces>On All Keypoints
Utility Menu>List>Loads>Forces>On Picked KPs

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLIST,*NODE1*,*NODE2*,*NINC*

Lists force loads on the nodes.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2*, *NINC

List forces for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If ALL, list for all selected nodes (**NSEL (p. 1266)**) and *NODE2* and *NINC* are ignored (default). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

Notes

Listing applies to the selected nodes (**NSEL (p. 1266)**) and the selected force labels (**DOFSEL (p. 569)**).

Caution:

A list containing a node number that is larger than the maximum defined node (*NODE2*), could deplete the system memory and produce unpredictable results.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Forces>On All Nodes

Utility Menu>List>Loads>Forces>On Picked Nodes

FLST,*NFIELD*,*NARG*,*TYPE*,*Otype*,*LENG*

Specifies data required for a picking operation (GUI).

DATABASE (p. 13): Picking (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NFIELD

Field number on the command which uses the picking data. (Count the command name as a field, so that a 2 indicates the first command argument, 3 for the second command argument, etc.) The corresponding field on the command will have a P51X label.

NARG

Number of items in the picked list.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TYPE

Type of items picked:

1

Node numbers

2

Element numbers

3

Keypoint numbers

4

Line numbers

5

Area numbers

6

Volume numbers

7

Trace points

8

Coordinate locations (in Global Cartesian coordinates)

9

Screen picks (in X, Y screen coordinates (-1 to 1))

Otype

Data order:

NOOR

Data is not ordered (default).

ORDER

Data is in an ordered list (such as for the **E (p. 591)**,P51X and **A (p. 81)**,P51X commands, in which the order of the data items is significant for the picking operation).

LENG

Length of number of items describing the list (should equal *NARG* if *Otype* = NOOR; default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Specifies data required for the **FITEM (p. 758)** command during a picking operation. This is a command generated by the GUI and will appear in the log file (`Jobname.log`) if graphical picking is used. This command is *not* intended to be typed in directly in a session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

On the log file, **FLST (p. 766)** is always followed by one or more **FITEM (p. 758)** commands which, in turn, are followed by the Mechanical APDL command containing a P51X label in one of its fields. Do not edit this set of commands.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

FLUXV

Calculates the flux passing through a closed contour.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

FLUXV (p. 768) invokes a Mechanical APDL macro which calculates the flux passing through a closed contour (path) predefined by **PATH (p. 1354)**.

The calculated flux is stored in the parameter FLUX.

In a 2D analysis, at least two nodes must be defined on the path. In 3D, a path of nodes describing a closed contour must be specified (that is, the first and last node in the path specification must be the same).

A counterclockwise ordering of nodes on the **PPATH (p. 1463)** command gives the correct sign on flux.

Path operations are used for the calculations, and all path items are cleared upon completion.

This macro is available for vector potential formulations only.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>Path Flux

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLUREAD, --, *Fname*, *Ext*, *KDIM*, *KOUT*, *LIMIT*, *ListOpt*

Reads one-way Fluent-to-Mechanical APDL coupling data via a .cgns file with one-side fast Fourier transformation complex pressure peak value.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

--

Reserved.

Fname

File name and directory path of a one-way Fluent-to-Mechanical APDL coupling data file (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name. Defaults to jobname.

Ext

File name extension of the one-way Fluent-to-Mechanical APDL coupling data file. Defaults to .cgns).

KDIM

Interpolation data for mapping. A value of 0 (default) or 2 applies 2D interpolation (where interpolation occurs on a surface).

KOUT

Outside region results for mapping:

0

Use the value(s) of the nearest region point for points outside of the region. This behavior is the default.

1

Set results extrapolated outside of the region to zero.

LIMIT

Number of nearby nodes considered for mapping interpolation. Minimum = 5. Default = 20.

Lower values reduce processing time; however, some distorted or irregular meshes require a higher value in cases where three nodes are encountered for triangulation.

ListOpt

Type of items picked:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(blank)

No listing (default).

SOURCE

List the node coordinates and complex pressure values on the Fluent source side during the solution.

TARGET

List the node coordinates and complex pressure values on the mapped Mechanical APDL target side during the solution.

BOTH

List the node coordinates and complex pressure values on both the Fluent source side and the mapped Mechanical APDL target side during the solution.

Notes

The **FLUREAD (p. 769)** command reads one-way Fluent-to-Mechanical APDL coupling data from a `.cgns` file. The Fluent one-side fast Fourier transformation (FFT) peak complex pressure values are mapped to the Mechanical APDL structure model during the acoustic-structural solution at each FFT frequency.

The command can be used only for the model with the acoustic elements.

To apply complex pressure to the structure model, define the SURF154 surface element, then define the one-way coupling interface (**SF (p. 1733)**,,FSIN) on the element.

You can define the solving frequency range via the HARFRQ (p. 885) command. The solver selects the FFT frequencies between the beginning and ending frequencies. The number of substeps is determined by the number of FFT frequencies over the frequency range. The number of substeps defined via the **NSUBST (p. 1287)** command is overwritten.

For better mapping performance, consider the following:

- Calculations for out-of-bound points require much more processing time than do points that are within bounds.
- For each point in the structural destination mesh, the command searches all possible triangles in the Fluent source mesh to find the best triangle containing each point, then performs a linear interpolation inside this triangle. For faster and more accurate results, consider your interpolation method and search criteria carefully. (See *LIMIT*.)

It is possible to apply one-way coupling excitation to multiple frequencies. The one-side FFT peak complex pressure values are necessary to do so.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/FOCUS, *WN*, *XF*, *YF*, *ZF*, *KTRANS*

Specifies the focus point (center of the window).

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

XF, YF, ZF

Location of the object to be at the focus point (center of the window) in the global Cartesian coordinate system. If *XF* = AUTO, allow automatic location calculation. If *XF* = USER, use focus location of last display (useful when last display had auto focus).

KTRANS

Translate key:

0

Interpret numerical *XF*, *YF*, *ZF* values as described above.

1

Interpret *XF*, *YF*, *ZF* values as multiples of half-screens to translate from the current position in the screen coordinate system. Example: *XF* of 2.4 translates the display approximately 2.4 half-screens to the left in the screen X (horizontal) direction.

2

Interpret *XF*, *YF*, *ZF* values as multiples of half-screens to translate from the current position in the global Cartesian coordinate system. Example: *XF* of 1.5 translates the display approximately 1.5 half-screens in the global Cartesian X direction of the model.

Command Default

Focus location is automatically calculated to be at the geometric center of the object (modified for centering within the window, depending upon the view).

Notes

Specifies the location on (or off) the model which is to be located at the focus point (center of the window). For section and capped displays, the cutting plane is also assumed to pass through this location (unless the working plane is used via **/CPLANE (p. 426)**). See also **/AUTO (p. 207)** and **/USER (p. 2061)** commands.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Focus Point

FORCE, *Lab*

Selects the element nodal force type for output.

POST1 (p. 51): Controls (p. 51)
POST26 (p. 58): Controls (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Type of force to be associated with the force items:

TOTAL

Total forces (static, damping, and inertia).

STATIC

Static forces.

DAMP

Damping forces.

INERT

Inertia forces.

Command Default

Use the total forces.

Notes

FORCE (p. 772) selects the element nodal force type for output with the POST1 **PRESOL (p. 1481)**, **PLESOL (p. 1401)**, **PRRFOR (p. 1517)**, **NFORCE (p. 1216)**, **FSUM (p. 780)**, etc. commands, the POST26 **ESOL (p. 672)** command, and reaction force plotting (**/PBC (p. 1356)**). For example, **FORCE (p. 772)**,STATIC causes item F of the **PRESOL (p. 1481)** command to be the static forces for the elements processed. Element member forces (such as those available for beams and shells and processed by Item and Sequence number) are not affected by this command. The SMISC records extract the static force.

In a non-spectrum analysis that includes either contact or pretension elements in the model, the **PRRSOL (p. 1518)** command is valid with the **FORCE (p. 772)** command. Otherwise, the **PRRSOL (p. 1518)** command is not valid with **FORCE (p. 772)**. Use the **PRRFOR (p. 1517)** command, which provides the same functionality as **PRRSOL (p. 1518)**, instead.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the **FORCE (p. 772)** command prior to any load case operations (**LCOPER (p. 1026)**) to insure the correct element nodal force combinations.

In POST26, the **ESOL (p. 672)** data stored is based on the active **FORCE (p. 772)** specification at the time the data is stored. To store data at various specifications (for example, static and inertia forces), issue a **STORE (p. 1867)** command before each new specification.

The **FORCE (p. 772)** command cannot be used to extract static, damping, and inertial forces for MPC184 joint elements.

To retrieve the different force types, use the ***GET (p. 797)** command with *Entity*=ELEM and *Item1*=EFOR.

The **FORCE (p. 772)** command is not supported in a spectrum analysis. You can specify the force type directly on the combination method commands (*ForceType* on the **PSDCOM (p. 1527)**, **SRSS (p. 1845)**, **CQC (p. 433)**, etc. commands).

The **FORCE (p. 772)** command is not supported in a modal analysis.

Menu Paths

Main Menu>General Postproc>Options for Outp

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

Utility Menu>List>Results>Options

FORM, *Lab*

Specifies the format of the file dump.

AUX2 (p. 61): Binary File Dump (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Format:

RECO

Basic record description only (minimum output) (default).

TEN

Same as RECO plus the first ten words of each record.

LONG

Same as RECO plus all words of each record.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Basic record description (RECO).

Notes

Specifies the format of the file dump (from the **DUMP (p. 583)** command).

Menu Paths

Utility Menu>File>List>Binary Files

Utility Menu>List>Files>Binary Files

/FORMAT,*NDIGIT*,*Ftype*,*NWIDTH*,*DSIGNF*,*LINE*,*CHAR*,*EXPTYPE*

Specifies format controls for tables.

POST1 (p. 51): Listing (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NDIGIT

Number of digits (3 to 32) in first table column (usually the node or element number). Initially defaults to 7.

Ftype

FORTRAN format type:

G --

G*xx*.*yy* (default)

F --

F*xx*.*yy*

E --

E*xx*.*yy*

where *xx* and *yy* are described below.

NWIDTH

Total width (9 to 32) of the field (*xx* in *Ftype*). Default = 13.

DSIGNF

Number of digits after the decimal point (*yy* in F or E format) or number of significant digits in G format. Range is 2 to *xx*-7 for *Ftype* = G or E; and 0 to *xx*-4 for *Ftype* = F. Default = 5.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINE

Number of lines (11 minimum) per page. Default = *ILINE* or *BLINE* (**/PAGE (p. 1348)**).

CHAR

Number of characters (41 to 240, system-dependent) per line before wraparound. Default = *ICHAR* or *BCHAR* (**/PAGE (p. 1348)**).

EXPTYPE

Number of digits for the exponent (*Ftype* = G):

3 --

Three digits (default).

2 --

Two digits.

Command Default

The program determines the data format.

Notes

Specifies various format controls for tables printed with the POST1 **PRNSOL (p. 1505)**, **PRESOL (p. 1481)**, **PRETAB (p. 1490)**, **PRRSOL (p. 1518)**, **PRPATH (p. 1517)**, and **CYCCALC (p. 453)** commands. A blank (or out-of-range) field on the command retains the current setting. Issue **/FORMAT (p. 774)**,STAT to display the current settings. Issue **/FORMAT (p. 774)**,DEFA to reestablish the initial default specifications.

For the POST26 **PRVAR (p. 1523)** command, the *Ftype*, *NWIDTH*, and *DSIGNF* fields control the time output format.

When viewing integer output quantities, the floating point format may lead to incorrect output values for large integers. You should verify large integer output values via the ***GET (p. 797)** command whenever possible.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*FREE, *Name*, *VAL1*

Deletes a matrix or a solver object and frees its memory allocation.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name

Name of the matrix or solver object to delete. Use *Name* = ALL to delete all APDL Math matrices and solver objects. Use *Name* = WRK to delete all APDL Math matrices and solver objects that belong to a given workspace.

VAL1

If *Name* = WRK, *Val1* is the memory workspace number.

Notes

A **/CLEAR (p. 350)** command will automatically delete all the current APDL Math objects.

Menu Paths

This command cannot be accessed from a menu.

FREQ, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*, *FREQ8*, *FREQ9*

Defines the frequency points for the SV (p. 1886) vs. FREQ (p. 776) tables.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FREQ1, FREQ2, FREQ3, . . . , FREQ9

Frequency points for SV vs. FREQ tables. Values must be in ascending order. *FREQ1* should be greater than zero. Units are cycles/time.

Command Default

No frequency table.

Notes

Repeat the **FREQ (p. 776)** command for additional frequency points (100 maximum). Values are added after the last nonzero frequency. If all fields (*FREQ1* -- *FREQ9*) are blank, erase SV vs. FREQ tables.

Frequencies must be in ascending order.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spectral values are input with the **SV (p. 1886)** command and interpreted according to the **SVTYP (p. 1887)** command. Applies only to the SPRS (single-point) option of the **SPOPT (p. 1837)** command. See the **SPFREQ (p. 1828)** command for frequency input in MPRS (multi-point) analysis.

Use the **STAT (p. 1861)** command to list current frequency points.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Erase Table

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Freq Table

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Erase Table

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Freq Table

---

FRQSCL, *Scaling*

Turns on automatic scaling of the entire mass matrix and frequency range for modal analyses.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Scaling

Off

Do not use automatic scaling of the mass matrix and frequency range.

On

Use automatic scaling of the mass matrix and frequency range.

Command Default

Mechanical APDL uses automatic scaling if appropriate.

Notes

This command is available only for modal analyses using the Block Lanczos, PCG Lanczos, Supernode, Subspace, or Unsymmetric mode extraction method (**MODOPT (p. 1140)**,LANB, LANPCG, SNODE, SUBPS, or UNSYM).

Use this command to deactivate or force activation of automatic scaling of the entire mass matrix and frequency range for modal analyses where the entire mass matrix is significantly different (that is, orders of magnitude different) than the entire stiffness matrix (for example, due to the particular unit system being used). Where the mass matrix is significantly smaller compared to the stiffness matrix, the eigenvalues will tend to approach very large numbers (>10e12), making the selected mode-extraction method less efficient and more likely to miss modes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can force the entire mass matrix and frequency range to be scaled via **FRQSCL (p. 777)**,ON. Doing so brings the stiffness and mass matrices closer together in terms of orders of magnitude, improving efficiency and reducing the likelihood of missed modes. The resulting eigenvalues are then automatically scaled back to the original system. If you are using micro MKS units, where the density is typically very small compared to the stiffness, you may want to force scaling on.

If the stiffness and mass are on the same scale, **FRQSCL (p. 777)**,ON has no effect.

This command is not valid and has no effect when used with **MSAVE (p. 1183)**,ON in a modal analysis with the PCG Lanczos mode extraction method.

Menu Paths

This command cannot be accessed from a menu.

FSCALE, *RFACT*, *IFACT*

Scales force load values in the database.

SOLUTION (p. 38): FE Forces (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RFACT

Scale factor for the real component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

IFACT

Scale factor for the imaginary component. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

Notes

Scales force load (force, heat flow, etc.) values in the database. Scaling applies to the previously defined values for the selected nodes (**NSEL (p. 1266)**) and the selected force labels (**DOFSEL (p. 569)**). Issue **FLIST (p. 766)** command to review results. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note:

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

FSCALE (p. 778) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Forces

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Forces

---

FSSECT, *RHO*, *NEV*, *NLOD*, *KBR*

Calculates and stores total linearized stress components.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RHO

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use a suitably large number (see the *Mechanical APDL Theory Reference*) or use -1 for an axisymmetric straight section.

NEV

Event number to be associated with these stresses (defaults to 1).

NLOD

Loading number to be associated with these stresses (defaults to 1).

KBR

For an axisymmetric analysis (*RHO* $\neq$ 0):

0

Include the thickness-direction bending stresses

1

Ignore the thickness-direction bending stresses

2

Include the thickness-direction bending stress using the same formula as the Y (axial direction) bending stress. Also use the same formula for the shear stress.

Notes

Calculates and stores the total linearized stress components at the ends of a section path (**PATH (p. 1354)**) (as defined by the first two nodes with the **PPATH (p. 1463)** command). The path must be entirely within the selected elements (that is, there must not be any element gaps along the path). Stresses are stored according to the fatigue event number and loading number specified. Locations (one for each node) are associated with those previously defined for these nodes (FL) or else they are automatically defined. Stresses are separated into six total components (SX through SXZ) and six membrane-plus-bending (SX

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

through SXZ) components. The temperature at each end point and the current time are also stored along with the total stress components. Calculations are made from the stresses currently in the database (last **SET (p. 1724)** or **LCASE (p. 1019)** command). Stresses are stored as section coordinate components if axisymmetric or as global Cartesian coordinate components otherwise, regardless of the active results coordinate system (**RSYS (p. 1639)**). The FSLIST command may be used to list stresses. The FS command can be used to modify stored stresses. See also the **PRSECT (p. 1520)** and **PLSECT (p. 1440)** commands for similar calculations.

Menu Paths

Main Menu>General Postproc>Fatigue>Store Stresses>At Cross Sect

---

FSSPARM, *PORT1*, *PORT2*

Calculates reflection and transmission properties of a frequency selective surface.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | Enterprise | **PrepPost** | Solver | AS add-on

PORT1

Port number of input port. Defaults to 1.

PORT2

Port number of output port. Defaults to 1.

Notes

FSSPARM (p. 780) calculates reflection and transmission coefficients, power reflection and transmission coefficients, and return and insertion losses of a frequency selective surface.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Port>FSS Parameters

FSUM, *LAB*, *ITEM*

Sums the nodal force and moment contributions of elements.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LAB

Coordinate system in which to perform summation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(blank)

Sum all nodal forces in global Cartesian coordinate system (default).

RSYS

Sum all nodal forces in the currently active RSYS coordinate system.

ITEM

Selected set of nodes.

(blank)

Sum all nodal forces for all selected nodes (default), excluding contact elements.

CONT

Sum all nodal forces for contact nodes only.

BOTH

Sum all nodal forces for all selected nodes, including contact elements.

Notes

Sums and prints, in each component direction for the total selected node set, the nodal force and moment contributions of the selected elements attached to the node set. Selecting a subset of nodes (**NSEL (p. 1266)**) and then issuing this command will give the total force acting on that set of nodes (default), excluding surface-to-surface, node-to-surface, line-to-line, and line-to-surface contact elements (TARGE169, TARGE170, CONTA172, CONTA174, CONTA175, and CONTA177).

Setting *ITEM* = CONT sums the nodal forces and moment contributions of the selected contact elements (CONTA172, CONTA174, CONTA175, and CONTA177). Setting *ITEM* = BOTH sums the nodal forces for all selected nodes, including contact elements.

Nodal forces associated with surface loads are not included. The effects of nodal coupling and constraint equations are ignored. Moment summations are about the global origin unless another point is specified with the **SPOINT (p. 1836)** command. This vector sum is printed in the global Cartesian system unless it is transformed (**RSYS (p. 1639)**) and a point is specified with the **SPOINT (p. 1836)** command. By default, the sum is done in global Cartesian, and the resulting vector is transformed to the requested system.

The *LAB* = RSYS option transforms each of the nodal forces into the active coordinate system before summing and printing. The **FORCE (p. 772)** command can be used to specify which component (static, damping, inertia, or total) of the nodal load is to be used. This command output is included in the **NFORCE (p. 1216)** command.

The command should not be used with axisymmetric elements because it might calculate a moment where none exists. Consider, for example, the axial load on a pipe modeled with an axisymmetric shell element. The reaction force on the end of the pipe is the total force (for the full 360 degrees) at that location. The net moment about the centerline of the pipe would be zero, but the program would incorrectly calculate a moment at the end of the element as the force multiplied by the radius.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The command is not valid for elements that operate solely within the nodal coordinate system with 1D option activated and rotated nodes (**NROTAT (p. 1264)**).

Using FSUM (p. 780) with the NLGEOM (p. 1231) Command

If you have activated large deflection (**NLGEOM (p. 1231)**,ON), the **FSUM (p. 780)** command generates the following message:

```
Summations based on final geometry and
   will not agree with solution reactions.
```

The message warns that the moment summations may not equal the real moment reactions. When calculating moment summations, the **FSUM (p. 780)** command assumes that the summation of rotations applies; however, it does not apply for large rotations, which require pseudovector representation to sum the rotations.

In contrast, the results for force reactions will be correct because they depend upon linear displacement vectors (which can be added).

Using FSUM (p. 780) in a Spectrum or PSD Analysis (ANTYPE (p. 162), SPECTR)

When using **FSUM (p. 780)** in a spectrum analysis (after the combination file has been input through **/INPUT (p. 948)**,,MCOM and when **SPOPT (p. 1837)** has not been issued with *Elcalc* = YES during the spectrum analysis), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS (p. 1639)** does not apply. The forces are in the nodal coordinate systems, and the vector sum is always printed in the global coordinate system.

The spectrum analysis summation is available when the element results are written to the mode file, Jobname.mode (*MSUPkey* = Yes on the **MXPAND (p. 1203)** command).

Because modal displacements cannot be used to calculate contact element nodal forces, *ITEM* does not apply to spectrum and PSD analyses.

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Total Force Sum

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FTRAN

Transfers solid model forces to the finite element model.

SOLUTION (p. 38): Solid Forces (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Forces are transferred only from selected keypoints to selected nodes. The **FTRAN (p. 783)** operation is also done if the **SBCTRAN (p. 1647)** command is issued or automatically done upon initiation of the solution calculations (**SOLVE (p. 1822)**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Forces

Main Menu>Solution>Define Loads>Operate>Transfer to FE>Forces

FTYPE,*FileType*,*PresType*

Specifies the file type and pressure type for the subsequent import of source points and pressures.

/MAP (p. 63): Pressure Mapping (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FileType

Type of file from which the pressure data will be retrieved (no default):

CFXTBR

File from a CFX Transient Blade Row (TBR) analysis export.

CFDPOST

File from a CFD-Post BC Profile export.

FORMATTED

Formatted file.

CSV

Comma-Separated Values file.

PresType

Type of pressure data contained in the file:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Only real-valued pressures are on the file.

1

Real-valued and imaginary-valued pressures are on the file (default).

Notes

CFX Transient Blade Row files (*FileType* = CFXTBR) are obtained from the **Export Results** Tab in CFX-Pre, with **[Export Surface Name]: Option** set to `Harmonic Forced Response`.

CFD-Post files (*FileType* = CFDPOST) are obtained from the **Export** action in CFD-Post with **Type** set to `BC Profile`.

Formatted files (*FileType* = FORMATTED) contain the coordinates and pressure data in fixed-format columns in the order `x, y, z, pressure`. You may have other columns of data in the file which can be skipped over in the *Format* specifier on the **READ (p. 1578)** command, but the data must be in that order.

Comma-separated values files (*FileType* = CSV) contain the coordinates and pressure data in comma-separated fields. The data can be in any order, and other fields of data may also be present.

Menu Paths

This command cannot be accessed from a menu.

FVMESH, *KEEP*

Generates nodes and tetrahedral volume elements from detached exterior area elements (facets).

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEEP

Specifies whether to keep the area elements after the tetrahedral meshing operation is complete.

0

Delete area elements (default).

1

Keep area elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **FVMESH (p. 784)** command generates a tetrahedral volume mesh from a selected set of detached exterior area elements (facets). (Detached elements have no solid model associativity.) The area elements can be triangular-shaped, quadrilateral-shaped, or a mixture of the two.

The **FVMESH (p. 784)** command is in contrast to the **VMESH (p. 2140)** command, which requires a volume to be input.

The main tetrahedra mesher (**MOPT (p. 1151)**,VMESH,MAIN) is the only tetrahedra mesher that supports the **FVMESH (p. 784)** command. The alternate tetrahedra mesher (**MOPT (p. 1151)**,VMESH,ALTERNATE) does not support **FVMESH (p. 784)**. MESH200 elements do not support **FVMESH (p. 784)**.

Tetrahedral mesh expansion (**MOPT (p. 1151)**,TETEXPND,*Value*) is supported for both the **FVMESH (p. 784)** and **VMESH (p. 2140)** commands. Tet-mesh expansion is the only mesh control supported by **FVMESH (p. 784)**.

Triangle- or quadrilateral-shaped elements may be used as input to the **FVMESH (p. 784)** command. Where quadrilaterals are used, the default behavior is for the pyramid-shaped elements to be formed at the boundary when the appropriate element type is specified. See the **MOPT (p. 1151)**,PYRA command for details.

The **FVMESH (p. 784)** command does not support multiple "volumes." If you have multiple volumes in your model, select the surface elements for one "volume," while making sure that the surface elements for the other volumes are deselected. Then use **FVMESH (p. 784)** to generate a mesh for the first volume. Continue this procedure by selecting one volume at a time and meshing it, until all of the volumes in the model have been meshed.

If an error occurs during the meshing operation, the area elements are kept even if *KEEP* = 0.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Tet Mesh From>Area Elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

G Commands

GAUGE, *Opt*, *FREQ*

Gauges the problem domain for a magnetic edge-element formulation.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Opt

Type of gauging to be performed:

ON

Perform tree gauging of the edge values (default).

OFF

Gauging is off. (You must specify custom gauging via APDL specifications.)

STAT

Gauging status (returns the current *Opt* and *FREQ* values)

FREQ

The following options are valid when *Opt* = ON:

0

Generate tree-gauging information once, at the first load step. Gauging data is retained for subsequent load steps. (This behavior is the default.)

1

Repeat gauging for each load step. Rewrites the gauging information at each load step to accommodate changing boundary conditions on the AZ degree of freedom (for example, adding or deleting AZ constraints via the **D (p. 483)** or **CE (p. 297)** commands).

Notes

The **GAUGE (p. 787)** command controls the tree-gauging procedure required for electromagnetic analyses using an edge-based magnetic formulation (elements SOLID226, SOLID227, SOLID236 and SOLID237).

Gauging occurs at the solver level for each solution (**SOLVE (p. 1822)**). It sets additional zero constraints on the edge-flux degrees of freedom AZ to produce a unique solution; the additional constraints are removed after solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the *FREQ* option to specify how the command generates gauging information for multiple load steps.

Access the gauging information via the _TGAUGE component of gauged nodes. The program creates and uses this component internally to remove and reapply the AZ constraints required by gauging. If *FREQ* = 0, the _TGAUGE component is created at the first load step and is used to reapply the tree gauge constraints at subsequent load steps. If *FREQ* = 1, the tree-gauging information and the _TGAUGE component are generated at every load step

If gauging is turned off (**GAUGE (p. 787)**,OFF), you must specify your own gauging at the APDL level.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>Gauging

Main Menu>Solution>Load Step Opts>Magnetics>Options Only>Gauging

---

GCDEF,*Option*,*SECT1*,*SECT2*,*MATID*,*REALID*,*SECT1END*,*SECT2END*

Defines interface interactions between general contact surfaces.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Option to be performed.

(blank)

Retain the previous *Option* setting between *SECT1* and *SECT2*.

AUTO

Define auto asymmetric contact between surfaces *SECT1* and *SECT2*.

SYMM

Define symmetric contact between surfaces *SECT1* and *SECT2*.

ASYM

Define asymmetric contact with *SECT1* as the source (contact) surface and *SECT2* as the target surface.

EXCL

Exclude contact between surfaces *SECT1* and *SECT2*. *MATID*, *REALID*, *SECT1END*, and *SECT2END* are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELETE

Remove the given definition from the **GCDEF (p. 788)** table. *MATID*, *REALID*, *SECT1END*, and *SECT2END* are ignored.

Note that **GCDEF (p. 788)**,DELETE,ALL,ALL does not remove the entire **GCDEF (p. 788)** table; it merely removes any existing **GCDEF (p. 788)**,,ALL,ALL definitions, while leaving intact any existing **GCDEF (p. 788)** definitions that are more specific.

To remove the entire **GCDEF (p. 788)** table, issue **GCDEF (p. 788)**,DELETE,TOTAL.

It is good practice to list all definitions using **GCDEF (p. 788)**,LIST before and after a **GCDEF (p. 788)**,DELETE command.

LIST

List stored **GCDEF (p. 788)** data entries. *MATID* and *REALID* are ignored.

GCDEF (p. 788),LIST lists all defined interactions. **GCDEF (p. 788)**,LIST,*SECT1*,*SECT2* lists the entry for the specific *SECT1*/*SECT2* interaction. **GCDEF (p. 788)**,LIST,ALL,ALL lists only the ALL,ALL entry (if any).

TABLE

List interpreted general contact definitions in tabular format. *MATID* and *REALID* are ignored.

By default, rows/columns of the table that match neighboring rows/columns are compressed to simplify the table. Issue **GCDEF (p. 788)**,TABLE,TOTAL to list the entire **GCDEF (p. 788)** table without removal of duplicate rows and columns.

TABLESOL

List a table showing actual interactions considered during solution. This option is only available after the **SOLVE (p. 1822)** command. *MATID* and *REALID* are ignored.

The table shows MAT and REAL entries considered during the solution (actual contact may or may not have occurred). This is in contrast to **GCDEF (p. 788)**,TABLE, which shows the user specifications. For auto asymmetric contact, TABLESOL indicates which of the possible contact versus target surface combinations was considered.

SECT1, SECT2

Section numbers representing *contact* (*SECT1*) and *target* (*SECT2*) general contact surfaces (no defaults). (In most cases, the actual determination of contact versus target surfaces takes place during **SOLVE (p. 1822)**.)

A node component name is also valid input for *SECT1* and *SECT2*. The component name is not stored. Instead, the program loops through all valid section IDs found in the component and creates **GCDEF (p. 788)** entries for all possible SECT1/SECT2 combinations that result. These entries are reflected in the *Option* = LIST and TABLE output. Section IDs can be further controlled by adding an extension (_EDGE, _FACE, _VERT, _TOP, or _BOT) to the end of the component name. See Specifying General Contact Interactions Between Nodal Components in the *Contact Technology Guide* for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following labels are also valid input:

SELF

Self contact.

ALL

All general contact sections IDs.

ALL_EDGE

Section IDs of all CONTA177 general contact line elements (which may be on the edges of 3D solid and shell base elements, or on beam base elements).

ALL_FACE

Section IDs of all general contact elements on faces of solid or shell base elements (both top and bottom faces of shell elements).

ALL_VERT

Section IDs of all CONTA175 general contact vertex elements (which may be on convex corners of solid and shell base elements, and on endpoints of beam base elements).

ALL_TOP

Section IDs of all general contact elements on top faces of shell base elements, and faces of solid base elements.

ALL_BOT

Section IDs of general contact elements on bottom faces of shell base elements, and faces of solid base elements.

The ALL_ labels apply to all defined general contact element section IDs in the model without regard to the select status of the elements or attached nodes.

See *SECT1*/*SECT2* Interactions (p. 791) for a description of how the various inputs for *SECT1* and *SECT2* are interpreted.

MATID

Material ID number for general contact interaction properties at the *SECT1*/*SECT2* interface. If zero or blank, the previous setting of *MATID* for *SECT1*/*SECT2* (if any) is retained.

As an example, you could specify "always bonded" contact behavior at the interface by setting *MATID* to 2 and issuing the command **TB (p. 1899)**,INTER,2,,,ABOND.

The coefficient of friction MU is also defined by *MATID*. Since the default is *MATID* = 0, frictionless contact (MU = 0) is assumed by default.

REALID

Real constant ID number for general contact interaction properties at the *SECT1*/*SECT2* interface. If zero or blank, the previous setting of *REALID* for *SECT1*/*SECT2* (if any) is retained.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As an example, you could specify contact stiffness (FKN) = 10 at the interface by setting *REALID* to 14 and issuing the command **R (p. 1565)**,14,,,10.

SECT1END*, *SECT2END

Last section number in the range. For *Option*= LIST, TABLE, or TABLESOL, data entries are processed for *contact* section numbers in the range from *SECT1* to *SECT1END*, and *target* section numbers in the range from *SECT2* to *SECT2END*. *SECT1END* and *SECT2END* are ignored for all other *Option* labels.

Notes

GCDEF (p. 788) defines the interface interaction between general contact surfaces identified by *SECT1* and *SECT2*. **GCDEF (p. 788)** commands are order independent in most cases.

GCDEF (p. 788) definitions should be issued after **GCGEN (p. 792)**. They are saved in the database and are written to and read from `.cdb` files.

See General Contact in the *Contact Technology Guide* for more information on the overall procedure to create general contact.

***SECT1*/*SECT2* Interactions**

The most specific types of general contact definitions are those described below:

SECT1 = any valid general surface section ID and *SECT2* = any different valid general surface section ID: *Option*, *MATID*, and *REALID* apply to general surface interactions between *SECT1* and *SECT2*. This is one of the most specific types of general contact definitions and is never overridden.

SECT1 = any valid general surface section ID and *SECT2* = *SECT1*: *Option*, *MATID*, and *REALID* apply to general surface self contact interactions involving *SECT1*. This is one of the most specific types of general contact definitions and is never overridden.

The remaining general contact definition types can be overridden by the above two general contact definition types:

SECT1 = ALL and *SECT2* = ALL: *Option*, *MATID*, and *REALID* apply to all general surface interactions, except where overridden by a more specific **GCDEF (p. 788)** command.

SECT1 = ALL and *SECT2* = SELF or *SECT1* = SELF and *SECT2* = ALL: *Option*, *MATID*, and *REALID* apply to all general surface self contact interactions, except where overridden by a more specific **GCDEF (p. 788)** command.

SECT1 = ALL and *SECT2* = any valid general surface section ID: *Option*, *MATID*, and *REALID* apply to all general surface interactions that include *SECT2*, except where overridden by a more specific **GCDEF (p. 788)** command.

SECT1 = any valid general surface section ID and *SECT2* = ALL: *Option*, *MATID*, and *REALID* apply to all general surface interactions that include *SECT1*, except where overridden by a more specific **GCDEF (p. 788)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

GCGEN, *Option*, *FeatureANGLE*, *EdgeKEY*, *SplitKey*, *SelOpt*

Creates contact elements for general contact.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Option to be performed.

NEW

Create a new general contact definition. This option removes all existing general contact elements and generates new elements with new section IDs. Any existing **GCDEF (p. 788)** specifications, general contact **SECTYPE (p. 1697)**/**SECDATA (p. 1655)** data, and general contact element types are also removed. If no general contact elements or data exist, this option behaves the same as *Option* = UPDATE.

UPDATE

Generate general contact elements on newly added (or selected) base elements. Newly generated contact elements are assigned new Section IDs. Existing general contact elements remain with their previously assigned section IDs and element attributes. Existing **GCDEF (p. 788)** and **SECTYPE (p. 1697)**/**SECDATA (p. 1655)** general contact data are respected. (This is the default option.)

DELETE

Remove all existing general contact elements. Existing **GCDEF (p. 788)** specifications, general contact **SECTYPE (p. 1697)**/**SECDATA (p. 1655)** data, and general contact element types are also removed.

SELECT

Select all existing general contact elements.

FeatureANGLE

Angle tolerance for determining feature edges (*EdgeKEY*) and general surfaces (*SplitKey*). Default = 42 degrees.

EdgeKEY

Key that controls creation of general contact line and vertex elements (CONTA177, CONTA175) on base elements. Line elements are overlaid on feature edges of 3D deformable bodies and on perimeter edges of 3D shell structures; vertex elements are overlaid on convex corners of deformable

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bodies and shell structures (2D or 3D), and on endpoints of 3D beam structures. See Understanding *FeatureANGLE* (p. 794) for an explanation of the feature edge criteria.

0

Exclude all edges and vertices (default).

1

Include CONTA177 elements on feature edges only.

2

Include CONTA177 elements on shell perimeter edges only.

3

Include CONTA177 elements on feature edges and shell perimeter edges.

4

Include CONTA177 elements on feature edges and shell perimeter edges, and CONTA175 elements on vertices.

5

Include CONTA175 elements on vertices only.

EdgeKey > 0 is intended to add extra contact constraint between an edge or vertex of one surface and other 3D surfaces. Both edge and vertex contact always use the penalty method and only include structural degrees of freedom (UX, UY, UZ).

GCGEN (p. 792) always creates CONTA177 elements on base beam elements, regardless of the *EdgeKEY* setting.

SplitKey

Key that controls how section IDs and contact element type IDs are assigned to surfaces.

SPLIT

Assign a different section ID and contact element type ID for every general surface of the selected base elements (default). See Understanding *FeatureANGLE* (p. 794) for an explanation of the split criteria. Different section IDs are assigned to the top and bottom surfaces of 2D shell and 3D shell bodies. This allows different **GCDEF (p. 788)** specifications for different portions of the assembly.

PART

Assign a different section ID and contact element type ID for every general surface which covers a physical part. Compared to the SPLIT option, this option produces fewer unique section IDs, which can make it easier to specify interactions via **GCDEF (p. 788)**. However, it may also result in a less accurate and/or less efficient solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SelOpt

Key that controls which base elements are considered for general contact.

ATTACH

Use a recursive adjacency selection to obtain complete physical parts (default). This selection starts from the selected base elements, progressively adding elements adjacent to the faces of selected elements until the edge of a part is reached. Then general contact elements are generated.

SELECT

Use only the initially selected base elements to generate general contact elements.

Notes

GCGEN (p. 792) creates general contact elements on the exterior faces of selected base elements. The base elements can be 2D or 3D solids, 3D beams, 2D shells (top and bottom), or 3D shells (top and bottom). The contact element types can be CONTA172, CONTA174, CONTA175, and/or CONTA177, depending upon the types of base elements in the model and the specified **GCGEN (p. 792)** options. General contact elements are identified by a real constant ID equal to zero.

You can control contact interactions between specific general contact surfaces that could potentially be in contact by using the **GCDEF (p. 788)** command. See General Contact in the *Contact Technology Guide* for more information on the overall procedure to create general contact.

Understanding *FeatureANGLE*

The exterior facets of the selected base solid and shell elements are divided into subsets based on the angle between the normals of neighboring faces. On a flat or smooth surface, adjacent exterior element faces have normals that are parallel or nearly parallel; that is, the angle between the adjacent normals is near zero degrees.

When the angle between the normals of two adjacent faces is greater than *FeatureANGLE*, the two faces are considered to be on two separate surfaces (*SplitKey* = SPLIT). The edge between the faces may be convex or concave. A convex (or outside) edge is considered to be a feature edge and may be affected by the *EdgeKEY* setting. For more information, see Feature Angle (*FeatureANGLE*) in the *Contact Technology Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/GCMD, *WN*, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*, *Lab10*, *Lab11*, *Lab12*

Controls the type of element or graph display used for the GPLOT (p. 866) command.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which this command applies (defaults to 1)

Lab1, Lab2, Lab3, . . . , Lab12

Command labels (for example, **PLNSOL (p. 1425)**,S,X)

Notes

This command controls the type of element or graph display that appears when you issue the **GPLOT (p. 866)** command when the **/GTYPE (p. 883)**,,(ELEM or GRPH) entity type is active. If you have multiple plotting windows enabled, you can also use **/GCMD (p. 795)** to select one window when you wish to edit its contents.

For related information, see the descriptions of the **GPLOT (p. 866)** and **/GTYPE (p. 883)** commands in this manual.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Multi-Plot Contrls

/GCOLUMN, *CURVE*, *STRING*

Allows the user to apply a label to a specified curve.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CURVE

Curve number on which label will be applied (integer value between 1 and 10).

STRING

Name or designation that will be applied to the curve (8 characters max).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is used for an array parameter plot (a plot created by the ***VPLOT (p. 2146)** command). Normally the label for curve 1 is "COL 1", the label for curve 2 is "COL 2" and so on; the column number is the field containing the dependent variables for that particular curve. Issuing **/GCOLUMN (p. 795)**,*CURVE*, with no string value specified resets the label to the original value.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

GENOPT

Specifies "General options" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>General Options

GEOM, *K2D*, *NDIV*

Defines the geometry specifications for the radiation matrix calculation.

AUX12 (p. 62): Radiation Matrix Method (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

K2D

Dimensionality key:

0

3D geometry (default)

1

2D geometry (plane or axisymmetric)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NDIV

Number of divisions in an axisymmetric model. Used only with *K2D* = 1. Defaults to 0 (2D plane). The 2D model is internally expanded to a 3D model based on the number of divisions specified (6 ≤ *NDIV* ≤ 90). For example, *NDIV* of 6 is internally represented by six 60° sections.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Other Settings

GEOMETRY

Specifies "Geometry" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Solid Model

***GET**,*Par*,*Entity*,*ENTNUM*,*Item1*,*IT1NUM*,*Item2*,*IT2NUM*

Retrieves a value and stores it as a scalar parameter or part of an array parameter.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Par

The name of the resulting parameter. See ***SET (p. 1720)** for name restrictions.

Entity

Entity keyword. Valid keywords are NODE, ELEM, KP, LINE, AREA, VOLU, etc., as shown for *Entity* = in the tables below.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ENTNUM

The number or label for the entity (as shown for *ENTNUM* = in the tables below). In some cases, a zero (or blank) *ENTNUM* represents all entities of the set.

Item1

The name of a particular item for the given entity. Valid items are as shown in the *Item1* columns of the tables below.

IT1NUM

The number (or label) for the specified *Item1* (if any). Valid *IT1NUM* values are as shown in the *IT1NUM* columns of the tables below. Some *Item1* labels do not require an *IT1NUM* value.

Item2,IT2NUM

A second set of item labels and numbers to further qualify the item for which data are to be retrieved. Most items do not require this level of information.

Notes

***GET (p. 797)** retrieves a value for a specified item and stores the value as a scalar parameter, or as a value in a user-named array parameter. An item is identified by various keyword, label, and number combinations. Usage is similar to the ***SET (p. 1720)** command except that the parameter values are retrieved from previously input or calculated results.

Example 8: *GET (p. 797) Usage

***GET (p. 797)**,A,ELEM,5,CENT,X returns the centroid x location of element 5 and stores the result as parameter A.

***GET (p. 797)** command operations, and corresponding get functions, return values in the active coordinate system (**CSYS (p. 441)** for input data or **RSYS (p. 1639)** for results data) unless stated otherwise.

A *get function* is an alternative in-line function that can be used instead of the ***GET (p. 797)** command to retrieve a value. For more information, see Using In-line Get Functions in the *Ansys Parametric Design Language Guide*.

Both ***GET (p. 797)** and ***VGET (p. 2118)** retrieve information from the active data stored in memory. The database is often the source, and sometimes the information is retrieved from common memory blocks that the program uses to manipulate information. Although POST1 and POST26 operations use a `*.rst` file, ***GET (p. 797)** data is accessed from the database or from the common blocks. Get operations do not access the `*.rst` file directly. For repeated gets of sequential items, such as from a series of elements, see the ***VGET (p. 2118)** command.

Most items are stored in the database after they are calculated and are available anytime thereafter. Items are grouped according to where they are usually first defined or calculated. Preprocessing data will often not reflect the calculated values generated from section data. Do not use ***GET (p. 797)** to obtain data from elements that use calculated section data, such as beams or shells.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the value retrieved by ***GET (p. 797)** is a component name, the resulting character parameter is limited to 32 characters. If the component name is longer than 32 characters, the remaining characters are ignored.

Most of the *general* items listed below are available from all modules. Each of the sections for accessing ***GET (p. 797)** parameters are shown in the following order:

- *GET General Entity Items (p. 799)
- *GET Preprocessing Entity Items (p. 803)
- *GET Solution Entity Items (p. 829)
- *GET Postprocessing Entity Items (p. 832)

The ***GET (p. 797)** command is valid in any processor.

General Items

*GET General Entity Items

- Table 115: *GET General Items, Entity = ACTIVE (p. 799)
- Table 116: *GET General Items, Entity = CMD (p. 800)
- Table 117: *GET General Items, Entity = COMP (p. 800)
- Table 118: *GET General Items, Entity = GRAPH (p. 801)
- Table 119: *GET General Items, Entity = PARM (p. 802)
- Table 120: *GET General Items, Entity = ***TBTYPE*** (p. 803)

Table 115: *GET General Items, Entity = ACTIVE

Entity **= ACTIVE,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797), Par, ACTIVE, 0,** ***Item1**,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
INT		Current interactive key: 0=off, 2=on.
IMME		Current immediate key: 0=off, 1=on.
MENU		Current menu key: 0=off, 1=on.
PRKEY		Printout suppression status: 0=**/NOPR (p. 1251)**, 1=**/GOPR (p. 865)** or **/GO (p. 865)**
UNITS		Units specified by **/UNITS (p. 2054)** command: 0 = USER, 1 = SI, 2 = CGS, 3 = BFT, 4 = BIN, 5 = MKS, 6 = MPA, 7 = uMKS.
ROUT		Current routine: 0 = Begin level, 17 = PREP7, 21 = SOLUTION, 31 = POST1, 36 = POST26, 52 = AUX2, 53 = AUX3, 62 = AUX12, 65 = AUX15.
TIME	WALL,CPU	Current wall clock or CPU time. Current wall clock will continue to accumulate during a run and is *not* reset to zero at midnight.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = ACTIVE, *ENTNUM* = 0 (or blank)		
GET (p. 797), Par, ACTIVE, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
DBASE	LDATE	Date of first modification of any database quantity required for POST1 operation. The parameter returned is *Par* = YEAR*10000 + MONTH*100 + DAY.
DBASE	LTIME	Time of last modification of any database quantity required for POST1 operation. The parameter returned is *Par* = HOURS*10000 + MINUTES*100 + SECONDS.
REV		Minor release revision number (5.6, 5.7, 6.0 etc.). Letter notation (for example, 5.0A) is not included.
TITLE	0,1,2,3,4	**Item2:** START **IT2NUM:** *N* Current title string of the main title (*IT1NUM*=0 or blank) or subtitle 1, 2, 3, or 4 (*IT1NUM*=1,2,3, or 4). A character parameter of up to 8 characters, starting at position *N*, is returned.
JOBNAM		**Item2:** START **IT2NUM:***N* Current Jobname. A character parameter of up to 8 characters, starting at position *N*, is returned. Use ***DIM (p. 530)** and ***DO (p. 567)** to get all 32 characters.
PLATFORM		The current platform.
NPROC	CURR, MAX, MAXP	The number of processors being used for the current session, or the maximum total number of processors (physical and virtual) available on the machine, or the maximum number of physical processors available on the machine. This only applies to shared-memory parallelism.
NUMCPU		Number of distributed processes being used (distributed-memory parallel solution).

Table 116: *GET General Items, Entity = CMD

Entity = CMD, *ENTNUM* = 0 (or blank) **The following items are valid for all commands except star (*) commands and non-graphics slash (/) commands.**		
GET (p. 797), *Par*, CMD, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
STAT		Status of previous command: 0=found, 1=not found (unknown).
NARGS		Field number of last nonblank field on the previous command.
FIELD	2,3...*N*	Numerical value of the *N*th field on the previous command. Field 1 is the command name (not available)

Table 117: *GET General Items, Entity = COMP

Entity = COMP, *ENTNUM* = 0 (or blank)		
GET (p. 797), *Par*, COMP, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
NCOMP		Total number of components and assemblies currently defined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= COMP,** ***ENTNUM*** **=** ***n*** **(*n*th component)**		
GET (p. 797)**, **Par*, COMP, *n*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
NAME		Name of the *N*th item (component or assembly) in the list of components and assemblies. A character parameter is returned.
Entity **= COMP,** ***ENTNUM*** **=** ***Cname*** **(component or assembly name)**		
GET (p. 797)**, **Par*, COMP, *Cname*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
TYPE		Type of component *Cname:* 1=Nodes, 2=Elements, 6=Keypoints, 7=Lines, 8=Areas, 9=Volumes, 11-15=Subcomponents (11=subcomponent at level 1, 12=subcomponent at level 2, etc.).
NSCOMP		Number of subcomponents (for assemblies).
SNAME	*N*	Name of *N*th subcomponent of assembly *Cname*. A character parameter is returned.

Table 118: *GET General Items, Entity = GRAPH

Entity **=GRAPH,** ***ENTNUM*** **= N (window number)**		
GET (p. 797)**, **Par*, GRAPH, N, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
ACTIVE		**/WINDOW (p. 2177)** status: 0=off, 1=on.
ANGLE		**/ANGLE (p. 140)** *THETA* angle.
CONTOUR	*Name*	**/CONTOUR (p. 415)** value for *Name*, where *Name* = VMIN, VINC, or NCONT.
DIST		**/DIST (p. 536)** *DVAL* value.
DSCALE	DMULT	**/DSCALE (p. 575)** *DMULT* value.
EDGE		**/EDGE (p. 594)** *KEY* value.
FOCUS	X, Y, Z	**/FOCUS (p. 771)** *CIN*, *YF*, or *ZF* value.
GLINE		**/GLINE (p. 860)** *STYLE* value.
MODE		**/USER (p. 2061)** or **/AUTO (p. 207)** setting: 0=user, 1=auto.
NORMAL		**/NORMAL (p. 1253)** *KEY* value.
RANGE	XMIN, XMAX, YMIN, YMAX	**/WINDOW (p. 2177)** *XMIN*, *XMAX*, *YMIN* , or *YMAX* screen coordinates.
RATIO	X, Y	**/RATIO (p. 1572)** *RATOX* or *RATOY*value.
SSCALE	SMULT	**/SSCALE (p. 1847)** *SMULT* value.
TYPE		**/TYPE (p. 2034)** *Type* value.
VCONE	ANGLE	**/VCONE (p. 2078)** *PHI* angle.
VIEW	X, Y, Z	**/VIEW (p. 2131)** *XV*, *YV*, or *ZV* value.
VSCALE	VRATIO	**/VSCALE (p. 2160)** *VRATIO* value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **=GRAPH, *ENTNUM* = 0 (or blank)**		
GET (p. 797), *Par*, GRAPH, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
DISPLAY		**/SHOW (p. 1785)** *VECT* setting: 0=raster, 1=vector.
ERASE		**/ERASE (p. 653)** or **/NOERASE (p. 1249)** setting: 0=no erase, 1=erase.
NDIST		Largest nodal range for current model (DX, DY, or DZ of the model).
NUMBER		**/NUMBER (p. 1290)** *NKEY* value.
PLOPTS	*Name*	**/PLOPTS (p. 1435)** setting of *Name*, where *Name*=LEG1, LEG2, LEG3, INFO, FRAM, TITL, MINM, or VERS.
SEG		Segment capability of graphics driver: 0=no segments available, 1=erasable segments available, 2=non-erasable segments available.
SHRINK		**/SHRINK (p. 1794)** *RATIO* value.

Table 119: *GET General Items, Entity = PARM

Entity **= PARM, *ENTNUM* = 0 (or blank)**		
GET (p. 797), *Par*, PARM, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
MAX		Total number of parameters currently defined.
BASIC		Number of scalar parameters (excluding parameters beginning with an underscore _, array parameters, and character parameters).
LOC	*Num*	Name of the parameter at the *Num* location in the parameter table. A character parameter is returned.
Entity **= PARM, *ENTNUM* = Name (parameter name)**		
GET (p. 797), *Par*, PARM, *Name*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
TYPE		Parameter type: 0 = scalar, 1 = array, 2 = table, 3 = character scalar, 4 = character array, -1 = undefined
DIM	X(1), Y(2), Z(3), (4), (5)	Row (X or 1), Column (Y or 2), Plane (Z or 3), Book (4), or Shelf (5) dimension of array parameter.
CSYS		Local coordinate system number
VAR	1, 2, 3, 4, 5	Name of primary variables 1-5. Primary variable names are character strings.
If Item1 = DIM and ITNUM refers to a dimension that does not exist, the program assigns a value of -1 to *Par*. If Item1 = CSYS and no local coordinate system number was assigned to the array parameter (*Name*), the program assigns a value of ZERO to *Par*.		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= PARM,** ***ENTNUM*** **= Name (parameter name)**		
GET (p. 797)**, **Par**, PARM, **Name**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
If Item1 = VAR and if IT1NUM refers to a primary variable that does not exist, the program assigns a blank value to *Par*.		

Table 120: *GET General Items, Entity = *TBTYPE*

Entity **=** ***TBTYPE*** **,** ***ENTNUM*** **=** ***MatID*** **(where TBTYPE is the material table type as defined via the TB (p. 1899) command, such (ELASTIC, CTE, etc.), and** ***MatID*** **is the material ID) Evaluates a material property coefficient for a given set of input field variables.**		
***GET (p. 797)**, **Par**, **TBTYPE**, **MatID**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM**, **Fld1**, **Fld2**,...*		
Item1	**IT1NUM**	**Description**
TBEV: Material table evaluation for query at a given field variable	*SINDEX* = Subtable index (1 - max number of subtables)	**Item2**: *CINDEX* = Coefficient index
		IT2NUM: *N* = Number of field variables input
		Fld1, *Fld2*, . . . , : *Val* = Value of the field variable(s), entered in the same order specified via the **TBFIELD (p. 1966)** command(s)

Preprocessing Items

*GET Preprocessing Entity Items

- Table 121: *GET Preprocessing Items, Entity = ACTIVE (p. 804)
- Table 122: *GET Preprocessing items, Entity = AREA (p. 805)
- Table 123: *GET Preprocessing Items, Entity = AXIS (p. 806)
- Table 124: *GET Preprocessing Items, Entity = CDSY (p. 806)
- Table 125: *GET Preprocessing Items, Entity = CE (p. 807)
- Table 126: *GET Preprocessing Items, Entity = CMPB (p. 807)
- Table 127: *GET Preprocessing Items, Entity = CP (p. 808)
- Table 128: *GET Preprocessing Items, Entity = CSEC (p. 808)
- Table 129: *GET Preprocessing Items, Entity = ELEM (p. 809)
- Table 130: *GET Preprocessing Items, Entity = ETYP (p. 811)
- Table 131: *GET Preprocessing Items, Entity = GCN (p. 811)
- Table 132: *GET Preprocessing Items, Entity = GENB (p. 812)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Table 133: *GET Preprocessing Items, Entity = GENS (p. 814)
- Table 134: *GET Preprocessing Items, Entity = KP (p. 815)
- Table 135: *GET Preprocessing Items, Entity = LINE (p. 816)
- Table 136: *GET Preprocessing Items, Entity = LINK (p. 817)
- Table 137: *GET Preprocessing Items, Entity = MAT (p. 818)
- Table 138: *GET Preprocessing Items, Entity = MPLAB (p. 818)
- Table 139: *GET Preprocessing Items, Entity = NODE (p. 818)
- Table 140: *GET Preprocessing Items, Entity = OCEAN (p. 820)
- Table 141: *GET Preprocessing Items, Entity = OCZONE (p. 821)
- Table 143: *GET Preprocessing Items, Entity = PART (p. 822)
- Table 142: *GET Preprocessing Items, Entity = PIPE (p. 821)
- Table 144: *GET Preprocessing Items, Entity = RCON (p. 822)
- Table 145: *GET Preprocessing Items, Entity = REIN (p. 823)
- Table 146: *GET Preprocessing Items, Entity = SCTN (p. 823)
- Table 147: *GET Preprocessing Items, Entity = SECP (p. 824)
- Table 148: *GET Preprocessing Items, Entity = SHEL (p. 825)
- Table 149: *GET Preprocessing Items, Entity = TBFT (p. 826)
- Table 150: *GET Preprocessing Items, Entity = TBLAB (p. 827)
- Table 151: *GET Preprocessing Items, Entity = VOLU (p. 828)

Table 121: *GET Preprocessing Items, Entity = ACTIVE

Entity **= ACTIVE,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, ACTIVE, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
SEG		Segment capability of graphics driver: 0=no segments available, 1=erasable segments available, 2=non-erasable segments available.
CSYS		Active coordinate system.
DSYS		Active display coordinate system.
MAT		Active material.
TYPE		Active element type.
REAL		Active real constant set.
ESYS		Active element coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = ACTIVE, *ENTNUM* = 0 (or blank)**		
GET (p. 797)*, *Par*, ACTIVE, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
SECT		Active section.
CP		Maximum coupled node set number in the model (includes merged and deleted sets until compressed out).
CE		Maximum constraint equation set number in the model (includes merged and deleted sets until compressed out).

Table 122: *GET Preprocessing items, Entity = AREA

***Entity* = AREA, *ENTNUM* = *N* (area number)**		
GET (p. 797)*, *Par*, AREA, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
ATTR	*Name*	Number assigned to the attribute, *Name*, where *Name*=MAT, TYPE, REAL, ESYS, KB,KE,SECN, NNOD, NELM, or ESIZ. (NNOD=number of nodes, NELM=number of elements, ESIZ=element size.)
ASEL		Select status of area *N*: -1=unselected, 0=undefined, 1=selected. Alternative get function: ASEL(*N*).
NXTH		Next higher area number above *N* in selected set (or zero if none found).
NXTL		Next lower area number below *N* in selected set (or zero if none found).
AREA		Area of area *N*. (**ASUM (p. 203)** or **GSUM (p. 881)** must have been performed sometime previously with at least this area *N* selected).
LOOP	1,2,...,*I*	*Item2*: LINE, *IT2NUM*: 1,2,...,*p* Line number of position *p* of loop *I*
***Entity* = AREA, *ENTNUM* = 0 (or blank)**		
GET (p. 797)*, *Par*, AREA, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
NUM	MAX, MIN	Highest or lowest area number in the selected set.
NUM	MAXD, MIND	Highest or lowest area number defined.
COUNT		Number of areas in the selected set.
AREA		Combined areas (from last **ASUM (p. 203)** or **GSUM (p. 881)**).
VOLU		Combined volume of areas (from last **ASUM (p. 203)** or **GSUM (p. 881)**. For 3D area elements, thickness is determined from area attributes (**AATT (p. 85)**). For 2D elements, area attributes are ignored and unit thickness is assumed.
CENT	X, Y, Z	Centroid X, Y, or Z location of areas (from last **ASUM (p. 203)** or **GSUM (p. 881)**).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last **ASUM (p. 203)** or **GSUM (p. 881)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= AREA,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, AREA, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last **ASUM (p. 203)** or **GSUM (p. 881)**).
IPR	X, Y, Z	Principal moments of inertia (from last **ASUM (p. 203)** or **GSUM (p. 881)**).
IXV	X, Y, Z	Principal orientation X vector components (from last **ASUM (p. 203)** or **GSUM (p. 881)**).
IYV	X, Y, Z	Principal orientation Y vector components (from last **ASUM (p. 203)** or **GSUM (p. 881)**).
IZV	X, Y, Z	Principal orientation Z vector components (from last **ASUM (p. 203)** or **GSUM (p. 881)**).

Table 123: *GET Preprocessing Items, Entity = AXIS

Entity **= AXIS,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, AXIS, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
COUNT	--	Number of defined sections.
NUM	MAX	Largest section number defined.
Entity **= AXIS,** ***ENTNUM*** **=** ***ID*** **(axis section identifier)**		
GET (p. 797)**, **Par**, AXIS, **ID**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
TYPE	--	Section type, for *ID* -- **SECTYPE (p. 1697)** command (always AXIS for axis sections).
NAME	--	Name defined for the given section ID number.
DATA	*nnn*	Where *nnn* is the location in the **SECDATA (p. 1655)** command for the given section ID number.

Table 124: *GET Preprocessing Items, Entity = CDSY

Entity **= CDSY,** ***ENTNUM*** **=** ***N*** **(coordinate system number)**		
GET (p. 797)**, **Par**, CDSY, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
LOC	X, Y, Z	X, Y, or Z origin location in global Cartesian system.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle (in degrees) relative to the global Cartesian coordinate system.
ATTR	*Name*	Number assigned to *Name*, where *Name*=KCS, KTHET, KPHI, PAR1, or PAR2. The value -1.0 is returned for KCS if the coordinate system is undefined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = CDSY, *ENTNUM* = *N* (coordinate system number)		
***GET (p. 797)**, *Par*, CDSY, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
NUM	MAX	The maximum coordinate system number

Table 125: *GET Preprocessing Items, Entity = CE

Entity = CE, *ENTNUM* = *N* (constraint equation set)		
***GET (p. 797)**, *Par*, CE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
If N = 0, then		
MAX		Maximum constraint equation number
NUM		Number of constraint equations
If N > 0, then		
NTERM		Number of terms in this constraint equation
CONST		Constant term for this constraint equation
TERM	number	Item2 = NODE: Gives the node for this position in the constraint equation. Item2 = DOF: Gives the DOF number for this position in the constraint equation. (1–UX, 2–UY, 3–UZ, 4–ROTX, etc.) Item2 = COEF: Gives the coefficient for this position in the constraint equation.

Table 126: *GET Preprocessing Items, Entity = CMPB

Entity = CMPB, *ENTNUM* = *N* (composite beam section identification number)		
***GET (p. 797)**, *Par*, CMPB, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
COUNT		Number of defined sections. If Item1 = COUNT, then N is blank.
NUM	MAX	Largest section number defined. If IT1NUM = MAX, then N is blank.
EXIS		Returns a 1 if the section exists and if it is a CMPB section.
NAME		The 8-character section name defined via the **SECTYPE (p. 1697)** command.
One of the following: CBMX CBTE CBMD		Item2 = NTEM (the number of temperatures for **CBMX (p. 288)**, **CBTE (p. 289)**, or **CBMD (p. 287)** data).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= CMPB,** ***ENTNUM*** **=** ***N*** **(composite beam section identification number)**		
****GET (p. 797)*****,** ***Par*****, CMPB,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
One of the following: CBMX CBTE CBMD		Item2 = TVAL; IT2NUM = *nnn* where *nnn* is the temperature value (< = NTEM).
One of the following: CBMX CBTE CBMD	*nnn*	Item2 = TEMP; IT2NUM = *tval* Where *nnn* is the location in the **CBMX (p. 288)**, **CBTE (p. 289)**, or **CBMD (p. 287)** command for the given coefficient number, and *tval* is the temperature value.

Table 127: *GET Preprocessing Items, Entity = CP

Entity **= CP,** ***ENTNUM*** **=** ***N*** **(coupled node set)**		
****GET (p. 797)*****,** ***Par*****, CP,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
If N = 0, then		
MAX		Maximum coupled set number
NUM		Number of coupled sets
If N > 0, then		
DOF		The degree of freedom for this set (1–UX, 2–UY, 3–UZ, 4–ROTX, etc.)
NTERM		Number of nodes in this set.
TERM	number	Item2 = NODE: Gives the node for this position number in the coupled set.

Table 128: *GET Preprocessing Items, Entity = CSEC

Entity **= CSEC,** ***ENTNUM*** **= 0 (or blank)**		
****GET (p. 797)*****,** ***Par*****, CSEC, 0,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
COUNT	--	Number of defined sections.
NUM	MAX	Largest section number defined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= CSEC,** ***ENTNUM*** **=** ***ID*** **(contact section identifier)**		
GET (p. 797),** ***Par**, CSEC,** ***ID*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
TYPE	--	Section type, for *ID* -- **SECTYPE (p. 1697)** command (always CONT for contact sections).
NAME	--	Name defined for the given section ID number.
DATA	*nnn*	Where *nnn* is the location in the **SECDATA (p. 1655)** command for the given section ID number.

Table 129: *GET Preprocessing Items, Entity = ELEM

Entity **= ELEM,** ***ENTNUM*** **=** ***N*** **(element number)**		
GET (p. 797),** ***Par**, ELEM,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
NODE	1, 2, ... 20	Node number at position 1,2,... or 20 of element*N*. Alternative get function: NELEM(*n, npos*), where *npos* is 1,2,...20.
CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system. The original locations is used even if large deflections are active. Alternative get functions: CENTRX(*N*), CENTRY(*N*), and CENTRZ(*N*) always retrieve the element centroid in global Cartesian coordinates, and are determined from the selected nodes on the elements.
ADJ	1, 2, ... 6	Element number adjacent to face 1,2,...6. Alternative get function: ELADJ(*N*,*face*). Only elements (of the same dimensionality) adjacent to lateral faces are considered.
ATTR	*Name*	Number assigned to the attribute *Name*, where *Name* = MAT, TYPE, REAL, ESYS, PSTAT, LIVE, or SECN. Returns a zero if the element is unselected. If *Name* = LIVE, returns a 1 if the element is selected and active, and a -1 if it is selected and inactive. *Name* = SECN returns the section number of the selected beam element. *Name* = ISOLID returns the underlying element number of the selected contact or target element. A **CNCHECK (p. 382)** command is needed before the ***GET (p. 797)** command.
LENG		Length of line element (straight line between ends).
LPROJ	X, Y, Z	Projected line element length (in the active coordinate system). X is x-projection onto y-z plane, Y is y projection onto z-x plane, and Z is z-projection onto x-y plane.
AREA		Area of area element.
APROJ	X, Y, Z	Projected area of area element area (in the active coordinate system). X is x-projection onto y-z plane, Y is y projection onto z-x plane, and Z is z-projection onto x-y plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= ELEM,** ***ENTNUM*** **=** ***N*** **(element number)**		
GET (p. 797)**, **Par**, ELEM, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
VOLU		Element volume. Based on unit thickness for 2D plane elements (unless the thickness option is used) and on the full 360 degrees for 2D axisymmetric elements. For general axisymmetric elements SOLID272 and SOLID273, only the area of the element on the master plane is reported before solving, not the volume. After solving, the volume is reported. **Note:** If results data are in the database, the volume returned is the volume calculated during solution.
ESEL		Select status of element *N*: -1 = unselected, 0 = undefined, 1 = selected. Alternative get function: ESEL(*N*).
NXTH		Next higher element number above *N* in selected set (or zero if none found). Alternative get function: ELNEXT(*N*)
NXTL		Next lower element number below *N* in selected set (or zero if none found).
HGEN		Heat generation on selected element *N*.
DGEN		Diffusing substance generation on selected node *N* (returns 0.0 if node is unselected, or if the DOF is inactive).
HCOE	face	Heat coefficient for selected element *N* on specified face. Returns the value at the first node that forms the face.
TBULK	face	Bulk temperature for selected element *N* on specified face. Returns the value at the first node that forms the face.
PRES	face	Pressure on selected element, *N* on specified face. Returns the value at the first node that forms the face.
SHPAR	*Test*	Element shape test result for selected element *N*, where *Test* = ANGD, ASPE (aspect ratio), JACR (Jacobian ratio), MAXA (maximum corner angle), PARA (deviation from parallelism of opposite edges), or WARP (warping factor).
MEMBER	COUNT	Number of reinforcing members (individual reinforcings) in the element *N*.
EGID	COUNT	Number of non-duplicate global identifiers in the element *N*.
	MIN, MAX	Lowest or highest global identifier in the element *N*.
Entity **= ELEM,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, ELEM, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NUM	MAX,MIN	Highest or lowest element number in the selected set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= ELEM,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, ELEM, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NUM	MAXD, MIND	Highest or lowest element number defined.
COUNT		Number of elements in the selected set.
MATM		Highest material number that is referenced by an element.
TYPM		Highest element type number that is referenced by an element.
RELM		Highest real constant number that is referenced by an element.
ESYM		Highest element coordinate system number that is referenced by an element.
SECM		Highest section ID number that is referenced by an element.
PRTM		Highest part number that is referenced by an element.
MEMBER	COUNT	Number of reinforcing members (individual reinforcings) in the selected set of reinforcing elements.
EGID	COUNT	Number of non-duplicate global identifiers in the selected set of reinforcing elements.
	MIN, MAX	Lowest or highest global identifier in the selected set of reinforcing elements.
	MIND, MAXD	Lowest or highest global identifier defined.

Table 130: *GET Preprocessing Items, Entity = ETYP

Entity **= ETYP,** ***ENTNUM*** **=** ***N*** **(element type number)**		
GET (p. 797)**, **Par**, ETYP, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
ATTR	*Name*	Number assigned to the attribute *Name*, where *Name*=ENAM, KOP1, KOP2, ..., KOP9, KO10, KO11, etc.
Entity **= ETYP,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**,**Par**,ETYP, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NUM	MAX	Maximum element type.

Table 131: *GET Preprocessing Items, Entity = GCN

Entity **= GCN,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, GCN, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
MAT	*Sect1*	*Item2* = 0 or blank; *IT2NUM* = *Sect2*. Material ID to be used for general contact between *Sect1* and *Sect2*. Alternative get function: SECTOMAT(*Sect1*,*Sect2*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = GCN, *ENTNUM* = 0 (or blank)**		
GET (p. 797)*, *Par*, GCN, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
REAL	*Sect1*	*Item2* = 0 or blank; *IT2NUM* = *Sect2*. Real constant ID to be used for general contact between *Sect1* and *Sect2*. Alternative get function: SECTOREAL(*Sect1*,*Sect2*).
DEF	*Sect1*	*Item2* = 0 or blank; *IT2NUM* = *Sect2*. Number indicating the type of contact for the general contact definition between *Sect1* and *Sect2*: = 0 - Excluded general contact between *Sect1* / *Sect2* = 1 - Asymmetric general contact between *Sect1* (contact) / *Sect2* (target) = 2 - Asymmetric general contact between *Sect1* (target) / *Sect2* (contact) = 3 - Symmetric general contact between *Sect1* / *Sect2*
Sect1 and *Sect2* are section numbers associated with general contact surfaces.		

Table 132: *GET Preprocessing Items, Entity = GENB

***Entity* = GENB, *ENTNUM* =*N* (nonlinear beam general section identification number)**		
GET (p. 797)*, *Par*, GENB, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
COUNT	(Blank)	Number of defined sections. If *Item1* = COUNT, then *N* is blank.
NUM	MAX	Largest section number defined. If *IT1NUM* = MAX, then *N* is blank.
EXIS	(Blank)	Returns a 1 if the section exists *and* if it is a GENB section.
SUBTYPE	(Blank)	Section subtype for the section ID specified via the **SECTYPE (p. 1697)** command.
NAME	(Blank)	The 8-character section name defined via the **SECTYPE (p. 1697)** command.
One of the following: BSAX BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE	(Blank)	*Item2* = NTEM, the number of temperatures for **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, or **BSTE (p. 278)** data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= GENB,** ***ENTNUM*** **=*N*** **(nonlinear beam general section identification number)**		
GET (p. 797),** ***Par**, GENB,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
One of the following: BSAX BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE	(Blank)	*Item2* = TVAL; *IT2NUM* = *nnn* Where *nnn* is the temperature value (<= NTEM).
One of the following: BSAX BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE	*nnn*	*Item2* = TEMP; *IT2NUM* = *tval* Where *nnn* is the location in the **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, or **BSTE (p. 278)** command for the given coefficient number, and *tval* is the temperature value. Examples for *nnn*: *nnn* = 1 for STRAIN(1) *nnn* = 2 for STRESS(1) *nnn* = 3 for STRAIN(2) *nnn* = 4 for STRESS(2) *nnn* = 5 for STRAIN(3) ...

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = GENB, *ENTNUM* =*N* (nonlinear beam general section identification number)		
***GET (p. 797)**, *Par*, GENB, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
One of the following: BSAX BSM1 BSM2 BSTQ BSS1 BSS2 BSMD BSTE	(Blank)	*Item2* = TEMP; *IT2NUM* = *tval*; *Item3* = NCONST The number of constants at *tval*.

Table 133: *GET Preprocessing Items, Entity = GENS

Entity = GENS, *ENTNUM* =*N* (preintegrated shell general section identification number)		
***GET (p. 797)**, *Par*, GENS, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
COUNT	(Blank)	Number of defined sections. If *Item1* = COUNT, then *N* is blank.
NUM	MAX	Largest section number defined. If *IT1NUM* = MAX, then *N* is blank.
EXIS	(Blank)	Returns a 1 if the section exists *and* if it is a GENS section.
NAME	(Blank)	The 8-character section name defined via the **SECTYPE (p. 1697)** command.
One of the following: SSPA SSPB SSPD SSPE SSMT SSBT SSPM	(Blank)	*Item2* = NTEM, the number of temperatures for **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, **SSBT (p. 1846)**, or **SSPM (p. 1855)** data.
One of the following: SSPA SSPB SSPD SSPE	(Blank)	*Item2* = TVAL; *IT2NUM* = *nnn* Where *nnn* is the temperature value (<= NTEM).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = GENS, *ENTNUM* =*N* (preintegrated shell general section identification number)**		
GET (p. 797)*, *Par*, GENS, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
SSMT SSBT SSPM		
One of the following: SSPA SSPB SSPD SSPE SSMT SSBT SSPM	*nnn*	*Item2* = TEMP; *IT2NUM* = *tval* Where *nnn* is the location in the **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, **SSBT (p. 1846)**, or **SSPM (p. 1855)** command for the given coefficient number, and *tval* is the temperature value.

Table 134: *GET Preprocessing Items, Entity = KP

***Entity* = KP, *ENTNUM* = *N* (keypoint number)**		
GET (p. 797)*, *Par*, KP, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system. Alternative get functions: KX(*N*), KY(*N*), KZ(*N*). Inverse get function: KP(*x, y, z*) returns the number of the selected keypoint nearest the *x, y, z* location (in the active coordinate system, lowest number for coincident keypoints).
ATTR	*Name*	Number assigned to the attribute *Name*, where *Name* = MAT, TYPE, REAL, ESYS, NODE, or ELEM.
KSEL		Select status of keypoint *N*: -1 = unselected, 0 = undefined, 1 = selected. Alternative get function: KSEL(*N*).
NXTH		Next higher keypoint number above *N* in selected set (or zero if none found). Alternative get function: KPNEXT(*N*).
NXTL		Next lower keypoint number below *N* in selected set (or zero if none found).
DIV		Divisions (element size setting) from **KESIZE (p. 978)** command.
***Entity* = KP, *ENTNUM* = 0 (or blank)**		
GET (p. 797)*, *Par*, KP, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
NUM	MAX, MIN	Highest or lowest keypoint number in the selected set.
NUM	MAXD, MIND	Highest or lowest keypoint number defined

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= KP,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, KP, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
COUNT		Number of keypoints in the selected set.
CENT	X, Y, Z	Centroid X, Y, or Z location of keypoints (from last **KSUM (p. 999)** or **GSUM (p. 881)**).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last **KSUM (p. 999)** or **GSUM (p. 881)**).
IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last **KSUM (p. 999)** or **GSUM (p. 881)**).
IPR	X, Y, Z	Principal moments of inertia (from last **KSUM (p. 999)** or **GSUM (p. 881)**).
IXV	X, Y, Z	Principal orientation X vector components (from last **KSUM (p. 999)** or **GSUM (p. 881)**).
IYV	X, Y, Z	Principal orientation Y vector components (from last **KSUM (p. 999)** or **GSUM (p. 881)**).
IZV	X, Y, Z	Principal orientation Z vector components (from last **KSUM (p. 999)** or **GSUM (p. 881)**).
MXLOC	X, Y, Z	Maximum X, Y, or Z keypoint coordinate in the selected set (in the active coordinate system).
MNLOC	X, Y, Z	Minimum X, Y, or Z keypoint coordinate in the selected set (in the active coordinate system).
NRELM	m	Keypoint number of meshed region nearest centroid of element *m*.

Table 135: *GET Preprocessing Items, Entity = LINE

Entity **= LINE,** ***ENTNUM*** **=** ***N*** **(line number)**		
GET (p. 797)**, **Par**, LINE, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
KP	1,2	Keypoint number at position 1 or 2.
ATTR	*Name*	Number assigned to the attribute, *Name*, where *Name*=MAT, TYPE, REAL, ESYS, NNOD, NELM, NDIV, NDNX, SPAC, SPNX, KYND, KYSP, LAY1, or LAY2. (NNOD=number of nodes, returns --1 for meshed line with no internal nodes, NELM=number of elements, NDIV=number of divisions in an existing mesh, NDNX=number of divisions assigned for next mesh, SPAC=spacing ratio in an existing mesh, SPNX=spacing ratio for next mesh, KYND=soft key for NDNX, KYSP=soft key for SPNX, LAY1=LAYER1 setting, LAY2=LAYER2 setting.)
LSEL		Select status of line *N*: -1=unselected, 0=undefined, 1=selected. Alternative get function: LSEL(*N*).
NXTH		Next higher line number above *N* in the selected set (or zero if none found). Alternative get function: LSNEXT(*N*)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= LINE,** ***ENTNUM*** **= *N* (line number)**		
GET (p. 797)**, **Par**, LINE, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NXTL		Next lower line number below *N* in selected set (or zero if none found).
LENG		Length. A get function LX(*n, lfrac*) also exists to return the X coordinate location of line *N* at the length fraction *lfrac* (0.0 to 1.0). Similar LY and LZ functions exist.
Entity **= LINE,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, LINE, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NUM	MAX, MIN	Highest or lowest line number in the selected set.
NUM	MIND, MAXD	Highest or lowest line number defined.
COUNT		Number of lines in the selected set.
LENG		Combined length of lines (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).
CENT	X, Y, Z	Centroid X, Y, or Z location of lines (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).
IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).
IPR	X, Y, Z	Principal moments of inertia (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).
IXV	X, Y, Z	Principal orientation X vector components (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).
IYV	X, Y, Z	Principal orientation Y vector components (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).
IZV	X, Y, Z	Principal orientation Z vector components (from last **LSUM (p. 1086)** or **GSUM (p. 881)**).

Table 136: *GET Preprocessing Items, Entity = LINK

Entity **= LINK,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, LINK, **NUM**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
COUNT		Number of defined sections.
NUM	MAX	Largest section number defined.
Entity **= LINK,** ***ENTNUM*** **= *id* (link section identification number)**		
GET (p. 797)**, **Par**, LINK, **id**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
TYPE		Section type (**SECTYPE (p. 1697)**) (always LINK for link sections) associated with the section identified via *id*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= LINK,** ***ENTNUM*** **=** ***id*** **(link section identification number)**		
****GET (p. 797)*****,** ***Par*****, LINK,** ***id*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
NAME		Name defined for the given section identification number
DATA	*nnn*	Location in the **SECDATA (p. 1655)** command for the given *id*
PROP	AREA	Area value
	ADDMAS	Added mass per unit length
	TENSKEY	Tension/compression key

Table 137: *GET Preprocessing Items, Entity = MAT

Entity **= MAT,** ***ENTNUM*** **= 0 (or blank)**		
****GET (p. 797)*****,** ***Par*****, MAT, 0,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
COUNT		Number of materials.
NUM	MAX	Largest material number for which at least one property is defined.

Table 138: *GET Preprocessing Items, Entity = MPLAB

Entity **= MPlab,** ***ENTNUM*** **=** ***N*** **(** ***MPlab*** **= material property label from MP (p. 1160) command;** ***N*** **= material number.)**		
****GET (p. 797)*****,** ***Par*****, MPlab,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
TEMP	val	Material property value at temperature of *val*. For temperature dependent materials, the program interpolates the property at temperature input for *val*.

Table 139: *GET Preprocessing Items, Entity = NODE

Entity **= NODE,** ***ENTNUM*** **=** ***N*** **(node number)**		
****GET (p. 797)*****,** ***Par*****, NODE,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
LOC	X, Y, Z	X, Y, Z location in the active coordinate system. Alternative get functions: NX(*N*), NY(*N*), NZ(*N*). Inverse get function. NODE(*x, y, z*) returns the number of the selected node nearest the *x, y, z* location (in the active coordinate system, lowest number for coincident nodes).
ANG	XY, YZ, ZX	THXY, THYZ, THZX rotation angle.
NSEL		Select status of node *N*: -1=unselected, 0=undefined, 1=selected. Alternative get function: NSEL(*N*).
NXTH		Next higher node number above *N* in selected set (or zero if none found). Alternative get function: NDNEXT(*N*).
NXTL		Next lower node number below *N* in selected set (or zero if none found).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= NODE,** ***ENTNUM*** **=** ***N*** **(node number)**		
GET (p. 797),** ***Par**, NODE,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
F	FX, MX, ...	Applied force at selected node *N* in direction *IT1NUM* (returns 0.0 if no force is defined, if node is unselected, or if the DOF is inactive). If *ITEM2* is IMAG, return the imaginary part. If the applied force is defined as a constant for this load step, regardless of the KBC setting, the result will be the force at the end time of a load step.
D	UX, ROTX, ...	Applied constraint force at selected node *N* in direction *IT1NUM* (returns a large number, such as 2e100, if no constraint is specified, if the node is unselected, or if the DOF is inactive). If *ITEM2* is IMAG, return the imaginary part. If the applied constraint force is defined as a constant for this load step, regardless of the KBC setting, the result will be the force at the end time of a load step.
HGEN		Heat generation on selected node *N* (returns 0.0 if node is unselected, or if the DOF is inactive).
NTEMP		Temperature on selected node N (returns 0.0 if node is unselected)
CPS	Lab	Couple set number with direction Lab = any active DOF, which contains the node *N*.
DGEN		Diffusing substance generation on selected node *N* (returns 0.0 if node is unselected, or if the DOF is inactive).
Entity **= NODE,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797),** ***Par**, NODE, 0,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
NUM	MAX, MIN	Highest or lowest node number in the selected set.
NUM	MAXD, MIND	Highest or lowest node number defined.
COUNT		Number of nodes in the selected set.
MXLOC	X, Y, Z	Maximum X, Y, or Z node coordinate in the selected set (in the active coordinate system).
MNLOC	X, Y, Z	Minimum X, Y, or Z node coordinate in the selected set (in the active coordinate system).
Note: If the application creates internal nodes during solution, the internal nodes will not be included. You can include them by using *KINTERNAL*. The command syntax is: ***GET (p. 797)**, *Par*, NODE, 0, Item1, IT1NUM, Item2, IT2NUM, KINTERNAL		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= NODE,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, NODE, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
The options for the *KINTERNAL* key are *(blank)*, which counts all nodes except internal nodes, and *INTERNAL*, which counts all nodes including internal nodes.		

Table 140: *GET Preprocessing Items, Entity = OCEAN

Entity **= OCEAN,** ***ENTNUM*** **=** ***Type*** **(where** ***Type*** **is a valid label on the** ***DataType*** **field of the OCTYPE (p. 1320) command)**		
GET (p. 797)**, **Par**, OCEAN, **Type**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NAME		Name defined for a given *Type*
DATA	1	Depth when *Type* = BASI
	2	Material ID when *Type* = BASI
	8	*KFLOOD* when *Type* = BASI
	9	*Cay* when *Type* = BASI
	10	*Cb* when *Type* = BASI
	11	*Zmsl* when *Type* = BASI
	13	*Caz* when *Type* = BASI
	14	*Ktable* when *Type* = BASI
	1	*KWAVE* when *Type* = WAVE
	2	*THETA* when *Type* = WAVE
	3	*WAVELOC* when *Type* = WAVE
	4	*KCRC* when *Type* = WAVE
	5	*KMF* when *Type* = WAVE
	6	*PRKEY* when *Type* = WAVE
PROP	NROW	Number of rows defined by **OCTABLE (p. 1316)** command
TABL	*i*	Data in table defined by **OCTABLE (p. 1316)** command *i* = row number; Item2 = column number

Entity **= OCEAN,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, OCEAN, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
COUNT		Number of defined global ocean entities (BASIC/CURRENT/WAVE)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= OCEAN,** ***ENTNUM*** **= 0 (or blank)**
****GET*** **(p. 797),** ***Par*****, OCEAN,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***
It may be necessary to determine *Cay*, *Caz*, *CMy*, or *CMz* during the solution process. In such cases, a negative value (-1.0) is returned to *Par* if the item is not specified in the database.

Table 141: *GET Preprocessing Items, Entity = OCZONE

Entity **= OCZONE,** ***ENTNUM*** **=** ***Name*** **(where** ***Name*** **is a valid label on the** ***ZoneName*** **field of the OCZONE (p. 1322) command)**		
****GET*** **(p. 797),** ***Par*****, OCZONE,** ***Name*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
DATA	8	*KFLOOD* for a given *ENTNUM* = *Name*
	9	*Cay* for a given *ENTNUM* = *Name*
	10	*Cb* for a given *ENTNUM* = *Name*
	13	*Caz* for a given *ENTNUM* = *Name*
PROP	NROW	Number of rows defined by **OCTABLE (p. 1316)** command
TABL	*i*	Data in table defined by **OCTABLE (p. 1316)** command *i* = row number; Item2 = column number
TYPE		Ocean zone type (returns 1, 2 or 3 for ZLOC-, COMP-, or PIP-type zones, respectively)
COMP		Component name when the given ocean zone type is COMP, or internal component name when the given ocean zone type is PIP
COMP2		External component name when the type of given ocean zone is PIP

Entity **= OCZONE,** ***ENTNUM*** **=** ***N***		
****GET*** **(p. 797),** ***Par*****, OCZONE,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
NAME		Name defined for a given zone ID (*ENTNUM*)

Entity **= OCZONE,** ***ENTNUM*** **= 0 (or blank)**		
****GET*** **(p. 797),** ***Par*****, OCZONE, 0,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
COUNT		Number of defined zones
If a requested item is not specified in the database, a negative value (-1.0) is returned to *Par*.		

Table 142: *GET Preprocessing Items, Entity = PIPE

Entity **= PIPE,** ***ENTNUM*** **= 0 (or blank)**		
****GET*** **(p. 797),** ***Par*****, PIPE,** ***NUM*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
COUNT		Number of defined sections
NUM	MAX	Largest section number defined

Entity **= PIPE,** ***ENTNUM*** **=** ***id*** **(pipe section identification number)**		
GET (p. 797),** ***Par**, PIPE,** ***id*****, Item1, IT1NUM, Item2, IT2NUM**		
Item1	**IT1NUM**	**Description**
TYPE		Section type, for id - **SECTYPE (p. 1697)** command (always PIPE for pipe sections)
SUBTYPE		Section type for id - **SECTYPE (p. 1697)** command
NAME		Name defined for the given section id number
DATA	*nnn*	Where *nnn* is the location in the **SECDATA (p. 1655)** command for the given section id number
SFLEX	*nnn*	Where *nnn* is the location in the **SFLEX (p. 1779)** command for the given section ID number
PROP	AREA	Area value
	IYY, IYZ, IZZ	Moments of inertia
	TORS	Torsion constant
	SCYY, SCYZ, SCZZ	Shear correction factors
	OFFY	Section offset in the Y direction.
	OFFZ	Section offset in the Z direction.
	ADDMAS	Added mass per unit length

Table 143: *GET Preprocessing Items, Entity = PART

Entity **= PART,** ***ENTNUM*** **=** ***N*** **(PART number)**		
GET (p. 797),** ***Par**, PART,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
TYPE		Element type number assigned to PART *N*.
MAT		Material number assigned to PART *N*.
REAL		Real constant number assigned to PART *N*.
Entity **= PART,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797),** ***Par**, PART, 0, Item1, IT1NUM, Item2, IT2NUM**		
NUMP		Total number of parts in the model.

Table 144: *GET Preprocessing Items, Entity = RCON

Entity **= RCON (p. 1574),** ***ENTNUM*** **=** ***N*** **(real constant set number)**		
GET (p. 797),** ***Par**, RCON,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
CONST	1, 2, ..., *m*	Value of real constant number *m* in set *N*.
***GET (p. 797),** *Par*, RCON, *0*, Item1, IT1NUM, Item2, IT2NUM		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = RCON (p. 1574), *ENTNUM* = *N* (real constant set number)		
*GET (p. 797), *Par*, RCON, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
NUM	MAX	The maximum real constant set number defined

Table 145: *GET Preprocessing Items, Entity = REIN

Entity = REIN, *ENTNUM* = *N* (reinforcing section identification number)		
*GET (p. 797), *Par*, REIN, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
TYPE		Section type, for ID -- **SECTYPE (p. 1697)** command (always REIN for reinforcing sections).
SUBTYPE		Section subtype for ID -- **SECTYPE (p. 1697)** command.
NAME		Name defined for a given ID number.
NREIN		Number of reinforcing fibers. For reinforcing sections generated (**EREINF (p. 656)**) via the standard method, the number of fibers defined via **SECDATA (p. 1655)**. For reinforcing sections generated (**EREINF (p. 656)**) via the mesh-independent method, the total number of fibers in the section.
TABL	*ReinfNum*, *I*	Reinforcing fiber data, as defined via **SECDATA (p. 1655)**. This item is not allowed for reinforcing sections generated (**EREINF (p. 656)**) via the mesh-independent method.

Table 146: *GET Preprocessing Items, Entity = SCTN

Entity = SCTN, *ENTNUM* = *N* (pretension section ID number)		
*GET (p. 797), *Par*, SCTN, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
1		Section ID number.
2		Section type (always 5 for pretension section).
3		Pretension node number.
4	Coordinate system number.	Section normal NX.
5	Coordinate system number.	Section normal NY.
6	Coordinate system number.	Section normal NZ.
7 or 8		Eight character section name.
9		Initial action key. Returns 0 or 1 for lock, 2 for "free-to-slide," or 3 for tiny.
10		Force displacement key. Returns 0 or 1 for force, or 2 for displacement.
11		First preload value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= SCTN,** ***ENTNUM*** **=** ***N*** **(pretension section ID number)**		
GET (p. 797)**, **Par**, SCTN, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
12		Load step in which first preload value is to be applied.
13		Load step in which first preload value is to be locked.
14...		14 through 17 is a repeat of 10 through 13, but for the second preload value; 18 through 21 is for the third preload value; and so forth.

Table 147: *GET Preprocessing Items, Entity = SECP

Entity **= SECP,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, SECP, **NUM**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
COUNT		Number of defined sections
NUM	MAX	Largest section number defined
Entity **= SECP,** ***ENTNUM*** **=** ***id*** **(beam section identification number)**		
***GET (p. 797)**, **Par**, SECP, **id**, Item1, IT1NUM, Item2, IT2NUM*		
Item1	**IT1NUM**	**Description**
TYPE		Section type, for id - **SECTYPE (p. 1697)** command (always BEAM for beam sections)
SUBTYPE		Section type for id - **SECTYPE (p. 1697)** command
NAME		Name defined for the given section id number
DATA	*nnn*	Where *nnn* is the location in the **SECDATA (p. 1655)** command for the given section id number
PROP	AREA	Area value
	IYY, IYZ, IZZ	Moments of inertia
	WARP	Warping constant
	TORS	Torsion constant
	CGY, CGZ	Y or Z coordinate center of gravity
	SHCY, SHCZ	Y or Z coordinate shear center
	SCYY, SCYZ, SCZZ	Shear correction factors
	OFFSET	Offset location: 1 = Centroid 2 = Shear Center 3 = Origin 0 = User Defined
	OFFY	Section offset in the Y direction.
	OFFZ	Section offset in the Z direction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = SECP, *ENTNUM* = *id* (beam section identification number)		
*GET (p. 797), *Par*, SECP, *id*, Item1, IT1NUM, Item2, IT2NUM		
Item1	**IT1NUM**	**Description**
	TXY	User transverse shear stiffness XY
	TXZ	User transverse shear stiffness XZ
	ADDMAS	Added mass per unit length

Table 148: *GET Preprocessing Items, Entity = SHEL

Entity = SHEL, ENTNUM = *N* (shell section identification number)		
*GET (p. 797), *Par*, SHEL, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
TYPE		Section type, for id -- **SECTYPE (p. 1697)** command. (always SHEL for shell sections)
NAME		Name defined for a given id number.
PROP	TTHK	Total thickness.
	NLAY	Number of layers.
	NSP	Number of section integration points.
	POS	Node position (as defined by **SECOFFSET (p. 1685)**).
		0 = User Defined.
		1 = Middle.
		2 = Top.
		3 = Bottom.
	OFFZ	User-defined section offset (POS = 0).
	TS11	Transverse shear stiffness factors.
	TS22	Transverse shear stiffness factors.
	TS12	Transverse shear stiffness factors.
	HORC	Homogeneous or complete section flag.
		0 = Homogeneous.
		1 = Composite.
	FUNC	Tabular function name for total thickness.
	UT11	User transverse shear stiffness 11.
	UT22	User transverse shear stiffness 22.
	UT12	User transverse shear stiffness 12.
	AMAS	Added mass.
	MSCF	Hourglass control membrane scale factor.
	BSCF	Hourglass control bending scale factor.
	DSTF	Drill stiffness scale factor.
	LDEN	Laminate density.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= SHEL, ENTNUM = *N* (shell section identification number)**		
GET (p. 797)*, *Par*, SHEL, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
	FKCN	*KCN* field value from the **SECFUNCTION (p. 1678)** command, in which the array or table is interpreted.
	ABD	Section membrane and bending stiffness matrix. Valid ITEM2 = 1,6 and IT2NUM = 1,6.
	E	Section transverse shear stiffness matrix. Valid ITEM2 = 1,2 and IT2NUM = 1,2.
LAYD	LayerNumber,THIC	Layer thickness.
	LayerNumber,MAT	Layer material.
	LayerNumber,ANGL	Layer orientation angle.
	LayerNumber,NINT	Number of layer integration points.

Table 149: *GET Preprocessing Items, Entity = TBFT

Entity **= TBFT, *ENTNUM* = *blank***		
GET (p. 797)*, *Par*, TBFT, , *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
nmat		Number of defined material models.
matnum	*index*	Material number in array (index varies for 1 to num materials).
Entity = TBFT, *ENTNUM* = *matid* (For getting names of constitutive function, matid = the material ID number)		
GET (p. 797)*, *Par*, TBFT, *matid*, *nfun*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	IT1NUM	Description
nfun		Number of constitutive functions for this material.
Entity = TBFT, *ENTNUM* = *matid* (To query constitutive function data, matid = the material ID number)		
GET (p. 797)*, *Par*, TBFT, *matid*, *func*, *fname*, *Item2*, *IT2NUM		
Item1	IT1NUM	Description
func	index	If Item2 = fname, the name of the constitutive function is returned.
func	function name	If Item2 = ncon, the number of constants is returned for the function specified in IT1NUM by the constitutive function name.
"	"	If Item2 = cons, set Item2num to index to return the value of the constant.
"	"	If Item2 = fixe, set Item2num to index to return the fix flag status.
"	"	If Item2 = RESI, returns the residual error while fitting the data.
"	"	If Item2 = type, returns the category of the constitutive model (moon, poly, etc.)
"	"	If Item2 = sord, returns the shear order of the prony visco model.
"	"	If Item2 = bord, returns the bulk order of the prony visco model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity* = TBFT, *ENTNUM* = *blank		
GET (p. 797)**, *Par*, TBFT, , *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
"	"	If Item2 = shif, returns the shift function name of the prony visco model.
Entity = TBFT, *ENTNUM* = *matid* (For getting names of experimental data, matid = the material ID number)		
GET (p. 797)**, *Par*, TBFT, *matid*, *nexp*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	IT1NUM	Description
nexp		Number of experiments for this material.
Entity = TBFT, *ENTNUM* = *matid* (To query experimental data, *matid* = the material ID number)		
GET (p. 797)**, *Par*, TBFT, *matid*, *func*, *fname*, *Item2*, *IT2NUM		
Item1	IT1NUM	Description
"	expindex	If Item2 = type, returns experiments type string.
"	"	If Item2 = numrow, returns number of rows in the data.
"	"	If Item2 = numcol, returns the number of cols in a row (set Intem2num = Row index)
"	"	If Item2 = data, returns the value of the data in *row, col* of exp expindex (set item2Num = row index and item3 = column index. All indices vary from 1 to the maximum value.
"	"	If Item2 = natt, returns the number of attributes.
"	"	If Item2 = attname, returns the attribute name (set Item2Num = Attr index).
"	"	If Item2 = attvald, returns double value of attribute (set Item2Num = Attr index).
"	"	If Item2 = attvali, returns integer valud of attribute (set Item2Num = Attr index).
"	"	If Item2 = attvals, returns the string value of the attribute (set Item2Num = Attr index).

Table 150: *GET Preprocessing Items, Entity = TBLAB

***Entity* = TBLAB, *ENTNUM* = *N*..(*TBlab* = data table label from the TB (p. 1899) command; *N* = material number.)**		
GET (p. 797)**, *Par*, *TBlab*, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*, *TBOPT		
Item1	**IT1NUM**	**Description**
TEMP	*T*	**Item2**: CONST **IT2NUM**: *Num* Value of constant number Num in the data table at temperature *T*. For constants, input an X,Y point;

Entity **= TBLAB,** ***ENTNUM*** **=** ***N*****..(*****TBlab*** **= data table label from the TB (p. 1899) command;** ***N*** **= material number.)**		
GET (p. 797),** ***Par**,** ***TBlab*****,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM*****,** ***TBOPT***		
Item1	**IT1NUM**	**Description**
		the constant numbers are consecutive with the X constants being the odd numbers, beginning with one.

Important:

To get all necessary output for materials defined via the **TB (p. 1899)** command, you must specify the final argument *TBOPT* as indicated in the syntax description above.

Table 151: *GET Preprocessing Items, Entity = VOLU

Entity **= VOLU,** ***ENTNUM*** **=** ***N*** **(volume number)**		
GET (p. 797),** ***Par**, VOLU,** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
ATTR	*Name*	Number assigned to the attribute *Name*, where *Name*=MAT, TYPE, REAL, ESYS, NNOD, or NELM. (NNOD=number of nodes, NELM=number of elements.)
VSEL		Select status of volume *N*: -1=unselected, 0=undefined, 1=selected. Alternative get function: VSEL(*N*).
NXTH		Next higher volume number above *N* in selected set (or zero if none found). Alternative get function: VLNEXT(*N*).
NXTL		Next lower volume number below *N* in selected set (or zero if none found).
VOLU		Volume of volume *N*. (**VSUM (p. 2167)** or **GSUM (p. 881)** must have been performed sometime previously with at least this volume *N* selected).
SHELL	1, 2, ..., *m*	**Item2**: AREA **IT2NUM**: 1,2,...,*p* Line number of position *p* of shell *m*
Entity = VOLU, *ENTNUM* = 0 (or blank)		
***GET (p. 797),** *Par*, VOLU, 0, Item1, IT1NUM, Item2, IT2NUM		
NUM	MAX, MIN	Highest or lowest volume number in the selected set.
NUM	MAXD, MIND	Highest or lowest volume number defined.
COUNT		Number of volumes in the selected set.
VOLU		Combined volumes (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).
CENT	X, Y, Z	Centroid X, Y, or Z location of volumes (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).
IOR	X, Y, Z, XY, YZ, ZX	Moments of inertia about origin (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IMC	X, Y, Z, XY, YZ, ZX	Moments of inertia about mass centroid (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).
IPR	X, Y, Z	Principal moments of inertia (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).
IXV	X, Y, Z	Principal orientation X vector components (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).
IYV	X, Y, Z	Principal orientation Y vector components (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).
IZV	X, Y, Z	Principal orientation Z vector components (from last **VSUM (p. 2167)** or **GSUM (p. 881)**).

Solution Items

*GET Solution Entity Items

- Table 152: *GET Solution Items, Entity = ACTIVE (p. 829)
- Table 153: *GET Solution Items, Entity = ELEM (p. 830)
- Table 154: *GET Solution Items, Entity = MODE (p. 831)
- Table 155: *GET Solution Items, Entity = DDAM (p. 832)

Table 152: *GET Solution Items, Entity = ACTIVE

Entity **= ACTIVE, *ENTNUM* = 0 (or blank)**		
GET (p. 797), *Par*, ACTIVE, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
ANTY		Current analysis type.
SOLU	DTIME	Time step size.
	NCMLS	Cumulative number of load steps.
	NCMSS	Number of substeps. NOTE: Used only for static, transient, and harmonic balance method (HBM) analyses.
	EQIT	Number of equilibrium iterations.
	NCMIT	Cumulative number of iterations.
	CNVG	Convergence indicator: 0=not converged, 1=converged.
	MXDVL	Maximum degree of freedom value.
	RESFRQ	Response frequency for 2nd order systems.
	RESEIG	Response eigenvalue for 1st order systems.
	DSPRM	Descent parameter.
	FOCV	Force convergence value.
	MOCV	Moment convergence value.
	HFCV	Heat flow convergence value.
	MFCV	Magnetic flux convergence value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = ACTIVE, *ENTNUM* = 0 (or blank)		
***GET (p. 797)**, *Par*, ACTIVE, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
	CSCV	Current segment convergence value.
	CUCV	Current convergence value.
	FFCV	Fluid flow convergence value.
	DICV	Displacement convergence value.
	ROCV	Rotation convergence value.
	TECV	Temperature convergence value.
	VMCV	Vector magnetic potential convergence value.
	SMCV	Scalar magnetic potential convergence value.
	VOCV	Voltage convergence value.
	PRCV	Pressure convergence value.
	VECV	Velocity convergence value.
	CRPRAT	Maximum creep ratio.
	PSINC	Maximum plastic strain increment.
	CGITER	Number of iterations in the PCG and symmetric JCG (non-complex version) solvers.

Table 153: *GET Solution Items, Entity = ELEM

Entity = ELEM, *ENTNUM* = 0 (or blank)		
***GET (p. 797)**, *Par*, ELEM, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
MTOT	X, Y, Z	Total mass components.
MC	X, Y, Z	Center of mass components.
IOR	X, Y, Z, XY, YZ, ZX	Moment of inertia about origin.
IMC	X, Y, Z, XY, YZ, ZX	Moment of inertia about the center of mass.
IPRIN	X, Y, Z	Principal moments of inertia.
IANG	XY, YZ, ZX	Angles of the moments of inertia principal axes.
FMC	X, Y, Z	Force components at mass centroid (1 (p. 830)).
MMOR	X, Y, Z	Moment components at origin (1 (p. 830)).
MMMC	X, Y, Z	Moment components at mass centroid (1 (p. 830)).

Note:

Items (1 (p. 830)) are available only after inertia relief solution (**IRLF (p. 960)**,1) or pre-calculation of masses (**IRLF (p. 960)**,-1).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item values are consistent with the mass summary printed in the output file. They are based on unscaled mass properties (see **MASCALE (p. 1110)** command).

Table 154: *GET Solution Items, Entity = MODE

***Entity* = MODE, *ENTNUM* = *N* (mode number)**		
GET (p. 797)**, *Par*, MODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
FREQ		Frequency of mode *N*. Returned values are valid for modal analyses which calculate real eigensolutions.
STAB		Stability value of mode *N*. Returned values are valid for modal analyses which calculate complex eigensolutions. The stability value is the real part of the complex eigenvalue. It contains information on the mode damping in a damped modal analysis.
DFRQ		Damped frequency of mode *N*. Returned values are valid for modal analyses which calculate complex eigensolutions. The damped frequency is the imaginary part of the complex eigenvalue.
PFACT		Participation factor of mode *N*. • If retrieved after a modal analysis, the real part of the participation factor is returned unless *IT1NUM* = IMAG. The direction is specified using *Item2* = DIREC and *IT2NUM* = X, Y, Z, ROTX, ROTY, or ROTZ • If retrieved after a spectrum analysis, the spectrum number M is specified using *Item2* = SPECT and *IT2NUM* = M. For a PSD analysis with spatial correlation or wave excitation, the retrieved participation factors will correspond to the first degree of freedom that is excited.
EFFM		Effective mass of mode *N*. Returned values are valid only after a modal analysis with effective mass calculation has been solved. The direction is specified using *Item2* = DIREC and *IT2NUM* = X, Y, Z, ROTX, ROTY, or ROTZ.
GENM		Generalized mass (also called modal mass) of mode *N*. Returned values are valid only after a modal analysis with generalized mass calculation has been solved.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = MODE, *ENTNUM* = *N* (mode number)		
***GET (p. 797)**, *Par*, MODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
MCOEF		Mode coefficient of mode *N*. Returned values are valid only after a spectrum analysis has been solved. The spectrum number M is specified using *Item2* = SPECT and *IT2NUM* = M. In a SPRS analysis, the values returned are based on the curve with the lowest damping. After a PSD analysis, the diagonal of the dynamic modal covariance matrix is retrieved for the displacement solution.
DAMP		Damping ratio of mode *N*. If retrieved after a modal analysis that creates complex solutions (DAMP, QRDAMP, or UNSYM eigensolvers) returned value is calculated from the complex frequencies. If retrieved after a spectrum analysis, returned value is the effective damping ratio. Not a function of direction. Also retrievable following a harmonic analysis or transient analysis with mode-superposition.

For all items except PFACT and MCOEF (as noted above), only the first 10000 values corresponding to significant modes will be returned.

The MODE file must be available to retrieve items PFACT and MCOEF with specified *Item2*. If *Item2* is not specified, the last calculated value will be returned.

All values retrieved correspond to the first load step values. For a Campbell diagram analysis (multistep modal), ***GET (p. 797)** with *Entity* = CAMP must be used.

Table 155: *GET Solution Items, Entity = DDAM

Entity = DDAM, *ENTNUM* = *N* (mode number)		
***GET (p. 797)**,*Par*,DDAM,*N*, *Item1*, *IT1NUM*		
Item1	**IT1NUM**	**Description**
DSHOCK		Shock design value of mode *N*.

If multiple DDAM analyses are performed, the last calculated value will be returned.

Postprocessing Items

*GET Postprocessing Entity Items

- Table 156: *GET Postprocessing Items, Entity = ACTIVE (p. 833)
- Table 157: *GET Postprocessing Items, Entity = ACUS (p. 834)
- Table 158: *GET Postprocessing Items, Entity = CAMP (p. 835)
- Table 159: *GET Postprocessing Items, Entity = CINT (p. 836)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Table 160: *GET Postprocessing Items, Entity = CYCCALC (p. 838)
- Table 161: *GET Postprocessing Items, Entity = ELEM (p. 838)
- Table 162: *GET Postprocessing Items, Entity = ETAB (p. 839)
- Table 163: *GET Postprocessing Items, Entity = FSUM (p. 840)
- Table 164: *GET Postprocessing Items, Entity = GSRESULT (p. 840)
- Table 165: *GET Postprocessing Items, Entity = MEMBER (p. 840)
- Table 166: *GET Postprocessing Items, Entity = NODE (p. 840)
- Table 167: *GET Postprocessing Items, Entity = PATH (p. 848)
- Table 168: *GET Postprocessing Items, Entity = PLNSOL (p. 849)
- Table 169: *GET Postprocessing Items, Entity = PRENERGY (p. 849)
- Table 170: *GET Postprocessing Items, Entity = PRERR (p. 850)
- Table 171: *GET Postprocessing Items, Entity = RAD (p. 850)
- Table 172: *GET Postprocessing Items, Entity = RSTMAC (p. 851)
- Table 173: *GET Postprocessing Items, Entity = SECR (p. 852)
- Table 174: *GET Postprocessing Items, Entity = SECTION (p. 855)
- Table 175: *GET Postprocessing Items, Entity = SORT (p. 856)
- Table 176: *GET Postprocessing Items, Entity = SSUM (p. 856)
- Table 177: *GET Postprocessing Items, Entity = VARI (p. 856)
- Table 178: *GET Postprocessing Items, Entity = XFEM (p. 857)

Table 156: *GET Postprocessing Items, Entity = ACTIVE

Entity **= ACTIVE,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**,**Par**, ACTIVE, 0,* ***Item1**,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
SET	LSTP	Current load step number.
	SBST	Current substep number.
	TIME	Time associated with current results in the database.
	FREQ	Frequency (for ANTYPE=MODAL, HARMIC, SPECTR; load factor for ANTYPE=BUCKLE).
	NSET	If Item2 is blank, number of data sets on result file. If Item2 = FIRST, IT2NUM = Loadstep, get set number of first substep of loadstep

Entity = ACTIVE, *ENTNUM* = 0 (or blank)		
*GET (p. 797),*Par*, ACTIVE, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
		If Item2 = LAST, IT2NUM = Loadstep, get set number of last substep of loadstep
RSYS		Active results coordinate system.

Table 157: *GET Postprocessing Items, Entity = ACUS

Entity = ACUS, *ENTNUM* = 0 (or blank)		
*GET (p. 797), *Par*, ACUS, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
PWL		Radiated sound power level
PRES		Far-field pressure at a given point
PHASE		Far-field pressure phase at a given point
SPL		Far-field sound pressure level at a given point
SPLA		Far-field a-weighted sound pressure level at a given point
DG		Far-field directivity at a given point
PS		Far-field scattered pressure at a given point
TS		Far-field target strength at a given point
DFIN		Diffuse sound field incident power
SIMP		Magnitude of specific acoustic impedance on the selected surface
AIMP		Magnitude of acoustic impedance on the selected surface
MIMP		Magnitude of mechanical impedance on the selected surface
APRES		Magnitude of average pressure on the selected surface
FORC		Magnitude of average force on the selected surface
POWER		Acoustic power through the selected surface
BSPL		SPL over frequency band
BSPA		A-weighted SPL over frequency band
PWRF		Reference sound power
TL		Transmission loss
RL		Return loss
Item1 = PWL, PRES, SPL, SPLA, PHASE, DG, PS, and TS are available after issuing the **PRFAR (p. 1490)** or **PLFAR (p. 1412)** command. The maximum values are obtained from the current command. Item1 = SIMP, AIMP, MIMP, APRES, FORC, POWER, TL, and RL are available after issuing the corresponding **PRAS (p. 1464)** command at the current frequency. The values are obtained at the current frequency, or at the last frequency for multiple load step and substep cases.		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = ACUS, *ENTNUM* = 0 (or blank)		
*GET (p. 797), *Par*, ACUS, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
Item1 = DFIN is available after the diffuse sound field solution.		

Table 158: *GET Postprocessing Items, Entity = CAMP

Note:

Available after PLCAMP (p. 1392) or PRCAMP (p. 1469) command is issued.

Entity = CAMP, *ENTNUM* = *N* (mode number)		
*GET (p. 797), *Par*, CAMP, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
NBMO		Number of modes in the Campbell diagram (*ENTNUM* not required). This value is the maximum value for *N*.
NBST		Number of steps in the Campbell diagram: modal load steps with rotational velocity (*ENTNUM* not required). This value is the maximum value for *M* (see Item1 = FREQ).
WHRL	*M*	Whirl of mode *N* at step *M*: -1 is backward whirl, 1 is forward whirl, and 0 is undetermined. For default *IT1NUM*, it corresponds to the whirl at the maximum rotational velocity.
VCRI		Critical speed for mode *N*. This value is available if an excitation is defined via the **PLCAMP (p. 1392)** or **PRCAMP (p. 1469)** command's *SLOPE* argument. (The unit of speed depends upon the *UNIT* value specified in those commands.) **Note:** N does not correspond to the mode number if *FREQB* (**PRCAMP (p. 1469)** or **PLCAMP (p. 1392)** command) is used. Instead, it represents the Nth mode number listed in the output of **PRCAMP (p. 1469)** or **PLCAMP (p. 1392)**.
FREQ	*M*	Natural frequency of mode (Hz) *N* at step *M*. It represents the complex part of the eigenvalue.
STAB	*M*	Stability value (Hz) of mode *N* at step *M*. It represents the real part of the eigenvalue.
UKEY	*M*	Instability key for mode *N* at step *M*: 0 is stable and 1 is unstable. For default *IT1NUM*, it corresponds to the stability over the whole rotational velocity range.
VSTA		Stability limit for mode N. This value is available when *SLOPE* is zero on the **PLCAMP (p. 1392)** or **PRCAMP (p. 1469)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Available after PLCAMP (p. 1392) or PRCAMP (p. 1469) command is issued.

Entity **= CAMP,** ***ENTNUM*** **=** ***N*** **(mode number)**		
GET (p. 797)**,**Par**, CAMP,* ***N**,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
		(The unit of speed depends upon the *UNIT* value specified in those commands.) **Note:** N does not correspond to the mode number if *FREQB* (**PRCAMP (p. 1469)** or **PLCAMP (p. 1392)** command) is used. Instead, it represents the Nth mode number listed in the output of **PRCAMP (p. 1469)** or **PLCAMP (p. 1392)**.

Note:

If the sorting is activated (Option=*ON* on the **PRCAMP (p. 1469)** and **PLCAMP (p. 1392)** commands), all the parameters retrieved are in the sorted order.

Table 159: *GET Postprocessing Items, Entity = CINT

Entity **= CINT,** ***ENTNUM*** **=** ***CrackId*** **(required Crack ID number)**		
GET (p. 797)**,**Par**, CINT,* ***CrackId**,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
CTIP	ctnum	*IT1NUM* = Crack tip node number (required) *Item2* = CONTOUR *IT2NUM* = Contour number (default 1) Returns JINT value if crack ID is JINT type; otherwise, returns 0. *Item1* defaults to CTIP, *Item2* defaults to CONTOUR.
Entity **= CINT,** ***ENTNUM*** **= CrackID (required Crack ID number)**		
GET (p. 797)**,* ***Par**, CINT,** ***CrackId*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM*****,** ***Item3*****,** ***IT3NUM*****,** ***Item4*****,** ***IT4NUM***		
Item1	**IT1NUM**	**Description**
CTIP	ctnum	*IT1NUM* = Crack tip node number (required)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= CINT,** ***ENTNUM*** **=** ***CrackId*** **(required Crack ID number)**		
****GET (p. 797)*****,*****Par*****, CINT,** ***CrackId*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
		Item2 = CONTOUR *IT2NUM* = Contour number (default 1) *Item3* = DTYPE *IT3NUM* = Data type (JINT, IIN1, IIN2, IIN3, K1, K2, K3, G1, G2, G3, GT, MFTX, MFTY, MFTZ, TSTRESS, CEXT, STTMAX, PSMAX, CSTAR, DLTA, DLTN, DLTK, KEQV, KANG, R, UFAC, CRDX, CRDY, CRDZ, and APOS) FOR *IT3NUM* = STTMAX or PSMAX: • Item4 = AINDEX (angle index) • IT4NUM = Index value (1 to N+1; N = Maximum number of intervals for the sweep (**CINT (p. 334)**,RSWEEP). Returns specified data type value. FOR *IT3NUM* = DLTA, DLTN, DLTK, KEQV, KANG, R, UFAC, CRDX, CRDY, CRDZ, APOS: • Set *IT2NUM* = 1 Returns specified data type value. *Item1* defaults to CTIP, *Item2* defaults to CONTOUR, *Item3* defaults to DTYPE. **Note:** DLTK in a 3D XFEM-based fatigue crack-growth analysis is evaluated based on the smoothed SIFS values. The actual DLTK value can be easily calculated by issuing ***GET (p. 797)** for SIFS values and the stress (load) ratio.
Entity **= CINT,** ***ENTNUM*** **= CrackID (required Crack ID number)**		
****GET (p. 797)*****,** ***Par*****, CINT,** ***CrackId*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM*****,**		
Item1	**IT1NUM**	**Description**
NNOD		Maximum number of nodes along the crack front.
Entity **= CINT,** ***ENTNUM*** **= CrackID (required Crack ID number)**		
****GET (p. 797)*****,** ***Par*****, CINT,** ***CrackId*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = CINT, *ENTNUM* = *CrackId* (required Crack ID number)		
*GET (p. 797),*Par*, CINT, *CrackId*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
NODE	ipos	IT1NUM = Position along the crack front (from 1 to NNOD). Default = 1. Returns node number at the given position along the crack front. (For XFEM, an internal node number is returned.)

Table 160: *GET Postprocessing Items, Entity = CYCCALC

Entity = CYCCALC, *ENTNUM* = CYCSPEC (p. 475) specification number Generate date for cyclic results using CYCCALC (p. 453) before retrieving those items.				
*GET (p. 797),*Par*,CYCCALC,*spec*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
FREQ	frequency point	SECTOR	sector	**CYCSPEC (p. 475)** result at the IT1NUM frequency and IT2NUM sector
		SECMAX	-	**CYCSPEC (p. 475)** maximum result at the IT1NUM frequency
		SECNUM	-	**CYCSPEC (p. 475)** sector with the maximum result at the IT1NUM frequency
		SECNODE	-	**CYCSPEC (p. 475)** node in the sector with the maximum result at the IT1NUM frequency
The frequency point refers to the harmonic solution data set number (NSET on the **SET (p. 1724)** command)				

Table 161: *GET Postprocessing Items, Entity = ELEM

Entity = ELEM (p. 618), *ENTNUM* = *N* (element number)		
*GET (p. 797),*Par*, ELEM, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
SERR[a]		Structural error energy.
SDSG[a]		Absolute value of the maximum variation of any nodal stress component.
TERR[a]		Thermal error energy.
TDSG[a]		Absolute value of the maximum variation of any nodal thermal gradient component.
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE.
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE.
KENE		Kinetic energy.
ASENE		Amplitude “stiffness” energy.
PSENE		Peak “stiffness” energy.
AKENE		Amplitude kinetic energy.
PKENE		Peak kinetic energy.
DENE		Damping energy.
WEXT[b]		Work due to external load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = **ELEM (p. 618)**, ***ENTNUM*** = ***N*** **(element number)**		
GET (p. 797)**,**Par**, **ELEM**, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
JHEAT		Element Joule heat generation (coupled-field calculation).
JS	X, Y, Z	Source current density (coupled-field calculation) in the global Cartesian coordinate system.
HS	X, Y, Z	Average element magnetic field intensity from current sources.
VOLU		Element volume, as calculated during solution.
ETAB	*Lab*	Value of element table item *Lab* for element *N* (see **ETABLE (p. 687)** command).
EFOR	*Nnum*	Element force at node *Nnum*. The force label is specified using Item2 = FX, FY, FZ, MX, MY, MZ, or HEAT. In a dynamics analysis, the element forces returned are based on the type of force requested. It is specified using the **FORCE (p. 772)** command for all dynamics analyses, except for spectrum analyses where *ForceType* is used on the combination commands (**SRSS (p. 1845)**, **PSDCOM (p. 1527)**, etc.).
SMISC	*Snum*	Value of element summable miscellaneous data at sequence number *Snum* (as used on **ETABLE (p. 687)** command).
NMISC	*Snum*	Value of element non-summable miscellaneous data at sequence number *Snum* (as used on **ETABLE (p. 687)** command).
FSOU		Element fluid flow source loading (poromechanics).

[a] Some element- and material-type limitations apply. For more information, see the documentation for the **PRERR (p. 1480)** command.

[b] WEXT is calculated for element-based loading only (and not for nodal-force loading). WEXT is stored on elements to which loading has been applied; if surface elements are added on top of other elements, for example, and pressure loading is applied to the surface elements, WEXT is available for the surface elements only.

Table 162: *GET Postprocessing Items, Entity = ETAB

Entity = **ETAB**, ***ENTNUM*** = ***N*** **(column number)**		
GET (p. 797)**,**Par**, **ETAB**, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
LAB		Label for column *N* of the element table (**ETABLE (p. 687)**). Returns a character parameter.
ELEM	*E*	Value in **ETABLE (p. 687)** column *N* for element number *E*.
Entity = **ETAB**, ***ENTNUM*** = **0 (or blank)**		
GET (p. 797)**,**Par**,**ETAB**,**0**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NCOL	MAX	Total number of **ETABLE (p. 687)** columns.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= ETAB,** ***ENTNUM*** **= 0 (or blank)**		
***GET (p. 797)**,**Par**,ETAB,0, Item1, IT1NUM, Item2, IT2NUM*		
Item1	**IT1NUM**	**Description**
NLENG	MAX	Largest element number defined.

Table 163: *GET Postprocessing Items, Entity = FSUM

Entity **= FSUM (p. 780),** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, **Par**, FSUM, 0, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
ITEM	*Lab*	Value of item *Lab* from last **FSUM (p. 780)** command. Valid labels are FX, FY, FZ, MX, MY, MZ, FLOW, HEAT, FLUX, etc.

Table 164: *GET Postprocessing Items, Entity = GSRESULT

Entity **= GSRESULT,** ***ENTNUM*** **= 0 (or blank) for generalized plane strain results in fiber direction**		
GET (p. 797)**, **Par**, GSRESULT, 0, **Item1**, **IT1NUM		
Item1	**IT1NUM**	**Description**
LFIBER		Fiber length change at ending point.
ROT	X,Y	Rotation angle of end plane about X or Y axis.
F		Reaction force at ending point.
M	X,Y	Reaction moment on ending plane.

Table 165: *GET Postprocessing Items, Entity = MEMBER

Entity **= MEMBER,** ***ENTNUM*** **=** ***N*** **(GroupID)**		
GET (p. 797)**,**Par**,MEMBER,**N**, **Item1**, **IT1NUM		
Item1	**IT1NUM**	**Description**
TEMP	MIN, MAX	Minimum or maximum temperature of members (individual reinforcings) with GroupID = *N* in the selected set of reinforcing elements.
Entity **= MEMBER,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**,**Par**,MEMBER,0, **Item1**, **IT1NUM		
Item1	**IT1NUM**	**Description**
TEMP	MIN, MAX	Minimum or maximum temperature in the selected set of reinforcing elements.

Table 166: *GET Postprocessing Items, Entity = NODE

Entity **=** ***NODE*****,** ***ENTNUM*** **=** ***N*** **(node number) for nodal degree-of-freedom results:**		
GET (p. 797)**, **Par**, NODE, **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum. Alternative get functions: UX(*N*), UY(*N*), UZ(*N*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for nodal degree-of-freedom results:**		
GET (p. 797)*, *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum. Alternative get functions: ROTX(*N*), ROTY(*N*), ROTZ(*N*).
TEMP		Temperature. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use TBOT, TE2, TE3, . . ., TTOP instead of TEMP. Alternative get functions: TEMP(*N*), TBOT(*N*), TE2(*N*), etc.
PRES		Pressure. Alternative get function: PRES(*N*).
GFV1, GFV2, GFV3		Nonlocal field values 1, 2, and 3.
VOLT		Electric potential. Alternative get function: VOLT(*N*).
CONC		Concentration.
MAG		Magnetic scalar potential. Alternative get function: MAG(*N*).
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum in a fluid analysis. X, Y, or Z nodal velocity or vector sum in a structural transient analysis (analysis with **ANTYPE (p. 162)**,TRANS). Alternative get functions: VX(*N*), VY(*N*), VZ(*N*).
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum in an electromagnetic analysis. X, Y, or Z nodal acceleration or vector sum in a structural transient analysis (analysis with **ANTYPE (p. 162)**,TRANS). Alternative get functions: AX(*N*), AY(*N*), AZ(*N*).
CURR		Current.
EMF		Electromotive force drop.
RF	FX, FY, FZ, MX, MY, MZ, CSGZ, BMOM, RATE, DVOL, FLOW, HEAT, AMPS or CHRG, FLUX, CURT, VLTG	Nodal reaction forces in the nodal coordinate system. The reaction forces returned are the total forces: static, plus damping, plus inertial, as appropriate based on analysis type (see **PRRSOL (p. 1518)** command). The first exception is modal analyses and mode-superposition transient analyses where static forces are returned. The second exception is spectrum analyses where the **PRRFOR (p. 1517)** logic is used internally. In this case, the reaction forces are based on the type of force requested (using *ForceType* with combination commands, such as **SRSS (p. 1845)**, **PSDCOM (p. 1527)**, etc.).
ORBT	A, B, PSI, PHI, YMAX, ZMAX, Whirl	Whirl orbit characteristics: A is the semi-major axis. B is the semi-minor axis. PSI is the angle between the local axis y and the major axis Y. PHI is the angle between initial position (t = 0) and major axis. YMAX is the maximum displacement along local y axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for nodal degree-of-freedom results:**		
GET (p. 797)*, *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
		ZMAX is the maximum displacement along local z axis. Whirl is the direction of an orbital motion (-1 is backward whirl, 1 is forward whirl, and 0 is undetermined). Angles PSI and PHI are in degrees and within the range of -180 through +180. Orbits are available only *after* issuing a **PRORB (p. 1515)** command.

Note:

Use this command carefully when N represents an internal node, as the nodal degrees of freedom may have different physical meanings.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for averaged nodal results based on selected elements:**			
GET (p. 797)*, *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM			
Item1	**IT1NUM**	**Description**	**Item2**
S	X, Y, Z, XY, YZ, XZ	Component stress.	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal stress.	
	INT, EQV	Stress intensity or equivalent stress.	
	MAXF	Maximum stress failure criterion.	
	TWSI	Tsai-Wu strength failure criterion.	
	TWSR	Inverse of Tsai-Wu strength ratio index failure criterion.	
EPTO	X, Y, Z, XY, YZ, XZ,	Component total strain (EPEL + EPPL + EPCR).	
	1, 2, 3	Principal total strain.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for averaged nodal results based on selected elements:**

GET (p. 797), *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM

Item1	IT1NUM	Description	Item2
	INT, EQV	Total strain intensity or total equivalent strain.	
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal elastic strain.	
	INT, EQV	Elastic strain intensity or elastic equivalent strain.	
	MAXF	Maximum strain failure criterion.	
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal plastic strain.	
	INT, EQV	Plastic strain intensity or plastic equivalent strain.	
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results.
	1, 2, 3	Principal creep strain.	
	INT, EQV	Creep strain intensity or creep equivalent strain.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for averaged nodal results based on selected elements:**

GET (p. 797), *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM

Item1	IT1NUM	Description	Item2
			ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
ESIG	X, Y, Z, XY, YZ, XZ	Components of Biot's effective stress.	
	1, 2, 3	Principal stresses of Biot's effective stress.	
	INT	Stress intensity of Biot's effective stress.	
	EQV	Equivalent stress of Biot's effective stress.	
CDM	DMG	Damage variable	
	LM	Maximum previous strain energy for virgin material	
GKS	X, XY, XZ	Gasket component stress.	
GKD	X, XY, XZ	Gasket component total closure.	
GKDI	X, XY, XZ	Gasket component total inelastic closure.	
GKTH	X, XY, XZ	Gasket component thermal closure.	
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal thermal strain.	
	INT, EQV	Thermal strain intensity or thermal equivalent strain.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for averaged nodal results based on selected elements:**

GET (p. 797), *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM

Item1	IT1NUM	Description	Item2
EPSW		Swelling strain.	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
FAIL	MAX	Maximum of all failure criterion defined for this node.	
	EMAX	Maximum Strain failure criterion.	
	SMAX	Maximum Stress failure criterion.	
	TWSI	Tsai-Wu Failure Criterion Strength Index failure criterion.	
	TWSR	Inverse of Tsai-Wu Strength Ratio Index failure criterion.	
	USR1, USR2, ..., USR9	User-defined failure criteria.	
	HFIB	Hashin Fiber Failure Criterion.	
	HMAT	Hashin Matrix Failure Criterion.	
	PFIB	Puck Fiber Failure Criterion.	
	PMAT	Puck Matrix Failure Criterion.	
	L3FB	LaRc03 Fiber Failure Criterion.	
	L3MT	LaRc03 Matrix Failure Criterion.	
	L4FB	LaRc04 Fiber Failure Criterion.	
	L4MT	LaRc04 Matrix Failure Criterion.	
NL	SEPL	Equivalent stress (from stress-strain curve).	
	SRAT	Stress state ratio.	
	HPRES	Hydrostatic pressure.	
	EPEQ	Accumulated equivalent plastic strain.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for averaged nodal results based on selected elements:**

GET (p. 797), *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM

Item1	IT1NUM	Description	Item2
	CREQ	Accumulated equivalent creep strain.	
	PSV	Plastic state variable or plastic work/volume.	
	PLWK	Plastic work/volume.	
TG	X, Y, Z, SUM	Component thermal gradient and sum. **Note:** IT1NUM = SUM is not supported for coupled pore-pressure-thermal (CPT*nnn*) elements.	
TF	X, Y, Z, SUM	Component thermal flux and sum. **Note:** IT1NUM = SUM is not supported for coupled pore-pressure-thermal (CPT*nnn*) elements.	
PG	X, Y, Z, SUM	Component pressure gradient and sum.	
EF	X, Y, Z, SUM	Component electric field and sum.	
D	X, Y, Z, SUM	Component electric flux density and sum.	
H	X, Y, Z, SUM	Component magnetic field intensity and sum.	
B	X, Y, Z, SUM	Component magnetic flux density and sum.	
CG	X, Y, Z, SUM	Component concentration gradient or vector sum.	
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum.	
FMAG	X, Y, Z, SUM	Component electromagnetic force and sum.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Entity* = *NODE*, *ENTNUM* = *N* (node number) for averaged nodal results based on selected elements:**

GET (p. 797), *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM

Item1	IT1NUM	Description	Item2
HS	X, Y, Z	Component magnetic field intensity from current sources (in the global Cartesian coordinate system).	
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution.	
FICT	TEMP	Fictive temperature.	
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; I1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.	
FFLX	X, Y, Z	Fluid flow flux in poromechanics.	
FGRA	X, Y, Z	Fluid pore pressure gradient in poromechanics.	
PMSV	VRAT, PPRE, DSAT, RPER	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure CPT elements.	
MPDP	TOTA, TENS, COMP, RW	Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).	
EPFR		Free strain in porous media	
DAMAGE	1,2,3,MAX	Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).	
GDMG		Damage	
IDIS		Structural-thermal dissipation rate	
CONT	STAT	Contact status.	
	PENE	Contact penetration.	
	PRES	Contact pressure.	
	SFRIC	Contact friction stress.	
	STOT	Contact total stress (pressure plus friction).	
	SLIDE	Contact sliding distance.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= *NODE*, *ENTNUM* = *N* (node number) for averaged nodal results based on selected elements:**			
GET (p. 797)*, *Par*, NODE, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM			
Item1	**IT1NUM**	**Description**	**Item2**
	GAP	Contact gap distance.	
	FLUX	Total heat flux at contact surface.	
	CNOS	Total number of contact status changes during substep.	
	FPRS	Actual applied fluid penetration pressure.	
SRES	SVAR*n*	Selected result *n*th state variable.	
	FLDUF0*n*	The *n*th user-defined field variable.	
Element nodal results are the average nodal value of the selected elements.			

Table 167: *GET Postprocessing Items, Entity = PATH

Entity = PATH, *ENTNUM* = 0 (or blank)		
GET (p. 797)*, *Par*, PATH, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
MAX	*Lab*	Maximum value of path item *Lab* from last path operation. Valid labels are the user-defined labels on the **PDEF (p. 1372)** or **PCALC (p. 1362)** command.
MAXPATH		Returns the maximum path number defined.
MIN	*Lab*	Minimum value of path item *Lab* from last path operation. Valid labels are the user-defined labels on the **PDEF (p. 1372)** or **PCALC (p. 1362)** command.
LAST	*Lab*	Last value of path item *Lab* from last path operation. Valid labels are the user-defined labels on the **PDEF (p. 1372)** or **PCALC (p. 1362)** command.
NODE		Value providing the number of nodes defining the path referenced in the last path operation.
ITEM	*Lab*	***Item2*** = PATHPT, ***IT2NUM*** = *n* The value of *Lab* at the *n*th data point from the last path operation.
POINT	*n*	***Item2*** = *X,Y,Z*, or *CSYS*. Returns information about the nth point on the current path.
NVAL		The number of path data points (the length of the data table) from the last path operation.
SET	*n*	***Item2*** = NAME. Returns the name of the *n*th data set on the current path.
NUMPATH		Returns the number of paths defined.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = PATH, *ENTNUM* = n (path number)		
Item1	**IT1NUM**	**Description**
NAME		Returns the name of the *n*th path.

Table 168: *GET Postprocessing Items, Entity = PLNSOL

Entity = PLNSOL (p. 1425)[a], *ENTNUM* = 0 (or blank)		
*GET (p. 797), *Par*, PLNSOL, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*		
Item1	**IT1NUM**	**Description**
MAX		Maximum value of item in last contour display (**PLNSOL (p. 1425)**, **PLESOL (p. 1401)**).
MIN		Minimum value of item in last contour display (**PLNSOL (p. 1425)**, **PLESOL (p. 1401)**).
BMAX		Maximum bound value of item in last contour display (**PLNSOL (p. 1425)**, **PLESOL (p. 1401)**).
BMIN		Minimum bound value of item in last contour display (**PLNSOL (p. 1425)**, **PLESOL (p. 1401)**).

[a] You must issue the **/SHOW (p. 1785)** command before commands that produce a graphical output when running in batch mode to produce/export graphic files. For more details, see External Graphics Options in the *Basic Analysis Guide*.

Table 169: *GET Postprocessing Items, Entity = PRENERGY

Note:

Available after the PRENERGY (p. 1478) command is issued.

Entity = PRENERGY, *ENTNUM* = *N* (component number)		
*GET (p. 797), *Par*, PRENERGY, *N*, *Item1*, *IT1NUM*		
Item1	**IT1NUM**	**Description**
NCMP		Number of components (*ENTNUM* not required). This value is the maximum value for *N*.
NENE		Number of energy types (*ENTNUM* not required). This value is the maximum value for *M*.
TOTE	*M*	Total energy of type *M* of the model (*ENTNUM* not required). Energy value is non-zero when *Cname1* is blank on prior **PRENERGY (p. 1478)** command.
ENG	*M*	Energy of type *M* of component *N*. Energy value is non-zero when *Cname1* is specified on prior **PRENERGY (p. 1478)** command.
PENG	*M*	Percentage of energy of type *M* of component *N*. Percentage of energy value is non-zero when *Cname1* is specified on prior **PRENERGY (p. 1478)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Available after the PRENERGY (p. 1478) command is issued.

Entity **= PRENERGY,** ***ENTNUM*** **= *N* (component number)**		
GET (p. 797)**, ***Par**, PRENERGY,** ***N*****,** ***Item1*****,** ***IT1NUM***		
Item1	**IT1NUM**	**Description**
Ordering of *N* and *M* corresponds to **PRENERGY (p. 1478)** output.		

Table 170: *GET Postprocessing Items, Entity = PRERR

Entity **= PRERR,** ***ENTNUM*** **= 0 (or blank)**		
GET (p. 797)**, ***Par**, PRERR, 0,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
SEPC[1]		Structural percent error in energy norm (**PRERR (p. 1480)**).
TEPC[1]		Thermal percent error in energy norm (**PRERR (p. 1480)**).
SERSM[1]		Structural error energy summation (**PRERR (p. 1480)**).
TERSM[1]		Thermal error energy summation (**PRERR (p. 1480)**).
SENSM[1]		Structural energy summation (**PRERR (p. 1480)**).
TENSM[1]		Thermal energy summation (**PRERR (p. 1480)**).

1. Some element- and material-type limitations apply. For more information, see the documentation for the **PRERR (p. 1480)** command.

Table 171: *GET Postprocessing Items, Entity = RAD

Entity **= RAD, ENTNUM = 0 (or blank)**		
GET (p. 797)**, ***Par**, RAD, 0,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
VFAVG		Value of the average view factor computed from the previous VFQUERY command.
Entity **= RAD, ENTNUM = n (enclosure number)**		
GET (p. 797)**, ***Par**, RAD,** ***n*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***		
Item1	**IT1NUM**	**Description**
NETHF		Value of the net heat flux lost by an enclosure. • For a perfect enclosure, it should be zero (due to numerical roundoff, it may be a small number close to zero).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= RAD, ENTNUM = n (enclosure number)**		
GET (p. 797)**, **Par**, RAD, **n**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
		• For an imperfect enclosure, it is the net heat escaping from the enclosure to the space node.

Table 172: *GET Postprocessing Items, Entity = RSTMAC

Note:

Available after RSTMAC (p. 1627) command is issued.

Entity **= RSTMAC (p. 1627), *ENTNUM*= *N* (solution number on *File1*)**

GET (p. 797)**, **Par**, RSTMAC, 0 or **N**, **Item1**, **IT1NUM**, **Item2**, **IT2NUM		
Item1	**IT1NUM**	**Description**
NS1		Total number of solutions (modes for example) read on *File1*. See *Sbstep1* on the **RSTMAC (p. 1627)** command. This value is the maximum value for *N*.
NS2		Total number of solutions (modes for example) read on *File2*. See *Sbstep2* on the **RSTMAC (p. 1627)** command. This value is the maximum value for *M*
MAC	M	Modal assurance criterion value (MAC) between the solution *N* read on *File1* and the solution *M* read on *File2*. **Note:** *N* and *M* do not correspond to the substep (or mode) numbers if NS1 and NS2 are different from the total number of substeps (or modes).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

Available after RSTMAC (p. 1627) command is issued.

Entity **= RSTMAC (p. 1627),** ***ENTNUM*** **=** ***N*** **(solution number on** ***File1*****)**

GET (p. 797),** ***Par**, RSTMAC, 0 or** ***N*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***

Item1	IT1NUM	Description
MACCYC	M	Modal assurance criterion value (MAC) from compressed table for **CYCLIC (p. 462)** between the solution *N* read on *File1* and the solution *M* read on *File2*. **Note:** *N* and *M* do not correspond to the substep (or mode) numbers if NS1 and NS2 are different from the total number of substeps (or modes).

Table 173: *GET Postprocessing Items, Entity = SECR

Entity **= SECR,** ***ENTNUM*** **=** ***n*** **(element number)**

For beam and pipe (including elbow) section results, return values for *all* elements

if the element number (*n*) is blank or ALL.

GET (p. 797),** ***Par**, SECR, n,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***

Item1	IT1NUM	Description	Item2	IT2NUM
S	X, Y, Z, XY, YZ, XZ	Component total stress	MAX – Returns maximum MIN – Returns minimum MAXY – Returns section Y location of maximum MAXZ – Returns section Z location of maximum MINY – Returns section Y location of minimum	These values apply only when Item2 = IVAL, JVAL, or KVAL. --- *For beam and pipe elements:* This value is the RST section-node number when KEYOPT(15) = 0. This value is the integration-point number when KEYOPT(15) = 1.
	1, 2, 3	Principal stress value		
	INT	Stress intensity value		
	EQV	Equivalent stress value		
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain		
	1, 2, 3	Principal total strain value		
	INT	Total strain intensity value		
	EQV	Equivalent total strain value		
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= SECR,** ***ENTNUM*** **=** ***n*** **(element number)**

For beam and pipe (including elbow) section results, return values for *all* elements

if the element number (*n*) is blank or ALL.

GET (p. 797), *Par*, SECR, n, *Item1*, *IT1NUM*, *Item2*, *IT2NUM

Item1	IT1NUM	Description	Item2	IT2NUM
	1, 2, 3	Principal elastic strain value	MINZ – Returns section Z location of minimum IVAL – Returns value at section node or integration point at element I node JVAL – Returns value at section node or integration point at element J node KVAL – Returns value at section integration point at element K node (ELBOW290) --- **For IVAL, JVAL, and KVAL:** The ALL (or blank) option for the element number is not valid. A specified element (**n**) is required.	*For the ELBOW290 element:* This value is the selected section-integration-station number. --- The numbers can be visualized via **SECPLOT (p. 1688)**, ,6.
	INT	Elastic strain intensity value		
	EQV	Equivalent elastic strain value		
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain		
	1, 2, 3	Principal thermal strain value		
	INT	Thermal strain intensity value		
	EQV	Equivalent thermal strain value		
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain		
	1, 2, 3	Principal plastic strain value		
	INT	Plastic strain intensity value		
	EQV	Equivalent plastic strain value		
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain		
	1, 2, 3	Principal component creep strain value		
	INT	Component creep strain intensity value		
	EQV	Equivalent component creep strain value		

Entity **= SECR,** ***ENTNUM*** **=** ***n*** **(element number)**

For beam and pipe (including elbow) section results, return values for *all* elements

if the element number (*n*) is blank or ALL.

GET (p. 797), *Par*, SECR, n, *Item1*, *IT1NUM*, *Item2*, *IT2NUM

Item1	IT1NUM	Description	Item2	IT2NUM
EPTT	X, Y, Z, XY, YZ, XZ	Component total mechanical *and* thermal and swelling strain		
	1, 2, 3	Principal total mechanical *and* thermal and swelling strain value		
	INT	Total mechanical *and* thermal and swelling strain intensity value		
	EQV	Equivalent total mechanical *and* thermal and swelling strain value		
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain		
	1, 2, 3	Principal diffusion strain value		
	INT	Diffusion strain intensity value		
	EQV	Equivalent diffusion strain value		
NL	SEPL	Plastic yield stress		
	SRAT	Plastic yielding (1 = actively yielding, 0 = not yielding)		
	HPRES	Hydrostatic pressure		
	EPEQ	Accumulated equivalent plastic strain		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= SECR,** ***ENTNUM*** **=** ***n*** **(element number)** **For beam and pipe (including elbow) section results, return values for** ***all*** **elements** **if the element number (*n*) is blank or ALL.**				
Par**, SECR, n,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***				
Item1	**IT1NUM**	**Description**	**Item2**	**IT2NUM**
	CREQ	Accumulated equivalent creep strain		
	PLWK	Plastic work/volume		
YSIDX	TENS,SHEA	Yield surface activity status: 1 for yielded and 0 for not yielded.		
FPIDX	TF01,SF01, TF02,SF02, TF03,SF03, TF04,SF04	Failure plane surface activity status: 1 for yielded and 0 for not yielded. Tension and Shear failure status are available for all four sets of failure planes.		

Table 174: *GET Postprocessing Items, Entity = SECTION

Entity **= SECTION,** ***ENTNUM*** **=** ***component*** **(listed below).** **Generate data for section stress results, using PRSECT (p. 1520) before retrieving these items. Valid labels for** ***ENTNUM*** **are MEMBRANE, BENDING, SUM (Membrane+Bending) , PEAK, and TOTAL. (The following items are not stored in the database and the values returned reflect the last quantities generated by PRSECT (p. 1520) or PLSECT (p. 1440).) Only MEMBRANE, BENDING, and SUM data are available after a PLSECT (p. 1440) command. The MEMBRANE label is only valid with** ***Item1*** **= INSIDE.**			
Par**, SECTION,** ***component*****,** ***Item1*****,** ***IT1NUM*****,** ***Item2*****,** ***IT2NUM***			
Item1	**IT1NUM**	**Item2**	**Description**
INSIDE	S	X, Y, Z, XY, YZ, XZ	Stress component at beginning of path.
		1, 2, 3	Principal stress at beginning of path.
		INT, EQV	Stress intensity or equivalent stress at beginning of path.
CENTER	S	X, Y, Z, XY, YZ, XZ	Stress component at midpoint of path.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= SECTION,*ENTNUM* = *component* (listed below).** **Generate data for section stress results, using PRSECT (p. 1520) before retrieving these items. Valid labels for *ENTNUM* are MEMBRANE, BENDING, SUM (Membrane+Bending) , PEAK, and TOTAL. (The following items are not stored in the database and the values returned reflect the last quantities generated by PRSECT (p. 1520) or PLSECT (p. 1440).) Only MEMBRANE, BENDING, and SUM data are available after a PLSECT (p. 1440) command. The MEMBRANE label is only valid with *Item1* = INSIDE.**			
GET (p. 797), *Par*, SECTION, *component*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM			
Item1	**IT1NUM**	**Item2**	**Description**
		1, 2, 3	Principal stress at midpoint of path.
		INT, EQV	Stress intensity or equivalent stress at midpoint of path.
OUTSIDE	S	X, Y, Z, XY, YZ, XZ	Stress component at end of path.
		1, 2, 3	Principal stress at end of path.
		INT, EQV	Stress intensity or equivalent stress at end of path.

Table 175: *GET Postprocessing Items, Entity = SORT

Entity **= SORT, *ENTNUM* = 0 (or blank)**		
GET (p. 797), *Par*, SORT, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
MAX		Maximum value of last sorted item (**NSORT (p. 1283)** or **ESORT (p. 680)** command).
MIN		Minimum value of last sorted item (**NSORT (p. 1283)** or **ESORT (p. 680)** command).
IMAX		Node/Element number where maximum value occurs.
IMIN		Node/Element number where minimum value occurs.

Table 176: *GET Postprocessing Items, Entity = SSUM

Entity **= SSUM, *ENTNUM* = 0 (or blank)**		
GET (p. 797), *Par*, SSUM, 0, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
ITEM	*Lab*	Value of item *Lab* from last **SSUM (p. 1858)** command. Valid labels are the user-defined labels on the **ETABLE (p. 687)** command.

Table 177: *GET Postprocessing Items, Entity = VARI

Entity **= VARI, *ENTNUM* = *N* (variable number after POST26 data storage) (for complex values, only the real part is returned with Item1 = EXTREM)**		
GET (p. 797),*Par*, *VARI*, *N*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM		
Item1	**IT1NUM**	**Description**
EXTREM	VMAX	Maximum extreme value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= VARI,** ***ENTNUM*** **= *N* (variable number after POST26 data storage) (for complex values, only the real part is returned with Item1 = EXTREM)**		
***GET (p. 797),Par,VARI,N,Item1,IT1NUM,Item2,IT2NUM**		
Item1	**IT1NUM**	**Description**
	TMAX	Time or frequency corresponding to VMAX.
	VMIN	Minimum extreme value.
	TMIN	Time or frequency corresponding to VMIN.
	VLAST	Last value.
	TLAST	Time or frequency corresponding to VLAST.
	CVAR	Covariance
REAL	f	Real part of variable *N* at time or frequency f.
IMAG	f	Imaginary part of variable *N* at frequency f.
AMPL	f	Amplitude value of variable *N* at frequency f
PHASE	f	Phase angle value of variable *N* at frequency f
RSET	*Snum*	Real part of variable *N* at location *Snum*.
ISET	*Snum*	Imaginary part of variable *N* at location *Snum*.
Entity **= VARI,** ***ENTNUM*** **= 0 (or blank) (after POST26 data storage)**		
***GET (p. 797),Par,VARI,0,Item1,IT1NUM,Item2,IT2NUM**		
Item1	**IT1NUM**	**Description**
NSETS		Number of data sets stored.

Table 178: *GET Postprocessing Items, Entity = XFEM

Entity **= XFEM,** ***ENTNUM*** **= 0 (or blank)**				
***GET (p. 797), Par, XFEM, 0, Item1, IT1NUM**				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
STAT	*Element Number*			Status of the element: 0 = uncracked, 1 = cracked
Entity **= XFEM,** ***ENTNUM*** **= NODE**				
***GET (p. 797), Par, XFEM, NODE, Item1, IT1NUM, Item2, IT2NUM**				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
ELEM	*Element Number*	POS (or blank)	1-4 (for 2D elements) 1-8 (for SOLID185)	Node number at position *IT2NUM* of cracked element

Entity = XFEM, ***ENTNUM*** = PHI				
GET (p. 797)**, **Par**, XFEM, PHI, **Item1**, **IT1NUM**, **Item2**, **IT2NUM				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
ELEM	*Element Number*	NODE	*Node Number*	LSM Phi value for this node number of this element
Entity = XFEM, ***ENTNUM*** = PSI				
GET (p. 797)**, **Par**, XFEM, PSI, **Item1**, **IT1NUM**, **Item2**, **IT2NUM				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
ELEM	*Element Number*	NODE	*Node Number*	LSM Psi value for this node number of this element

Menu Paths

Utility Menu>Parameters>Get Scalar Data

/GFILE, *SIZE*

Specifies the pixel resolution on Z-buffered graphics files.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIZE

Pixel resolution. Defaults to a pixel resolution of 800. Valid values are from 256 to 2400.

Command Default

800 pixels

Notes

Defines the pixel resolution on subsequently written graphics files (for example, JPEG, PNG, TIFF) for software Z-buffered displays (**/TYPE (p. 2034)**). Lowering the pixel resolution produces a "fuzzier" image; increasing the resolution produces a "sharper" image but takes a little longer.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/GFORMAT,*Ftype*,*NWIDTH*,*DSIGNF*

Specifies the format for the graphical display of numbers.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ftype

FORTRAN format types (G is the default if this field is left blank.)

G

G*xx*.*yy*. *xx* and *yy* are described below.

F

F*xx*.*yy*

E

E*xx*.*yy*

NWIDTH

Total width (12 maximum) of the field (the *xx* in *Ftype*). Defaults to 12.

DSIGNF

Number of digits after the decimal point (*yy* in F or E format) or number of significant digits in G format. Range is 1 to *xx*-6 for *Ftype* = G or E; and 0 to *xx*-3 for *Ftype* = F. The default is a function of *Ftype* and *NWIDTH*.

Notes

Enables you to control the format of the graphical display of floating point numbers.

To display the current settings, issue **/GFORMAT (p. 859)**,STAT..

To allow Mechanical APDL to select the format for the graphical display of floating numbers, issue **/GFORMAT (p. 859)**,DEFA.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Floating Point Format

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/GLINE, *WN*, *STYLE*

Specifies the element outline style.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

STYLE

Outline key:

0

Solid element outlines (default)

1

Dashed element outlines

-1

No element outlines

Command Default

Solid element outlines

Notes

Determines the element outline style. Often used when node numbers are displayed to prevent element lines from overwriting node numbers.

Unless you are using an OpenGL or Starbase driver, the dashed element outline option (**/GLINE (p. 860)**,*WN*,1) is not available in the following situations:

- Z-buffered displays (**/TYPE (p. 2034)**,*WN*,6).
- Capped Z-buffered displays (**/TYPE (p. 2034)**,*WN*,7).
- Qslice Z-buffered displays (**/TYPE (p. 2034)**,*WN*,8).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Edge Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/GMARKER, *CURVE*, *KEY*, *INCR*

Specifies the curve marking style.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CURVE

Curve number markers will be applied on (integer value between 1 and 10).

KEY

Marker key:

0

No markers will be applied (default).

1

TRIANGLES will be applied.

2

SQUARES will be applied.

3

DIAMONDS will be applied.

4

CROSSES will be applied.

INCR

Determines the curve marking frequency. (a whole number value between 1 and 255). If *INCR* = 1, markers are displayed at every data point on the curve. If *INCR* = 2 then markers are displayed at every second data point. If *INCR* = 3 then they are displayed at every third data point.

Command Default

No markers will be applied.

Notes

The user-specified markers will not be drawn when the area under the curve is color-filled (**/GROPT (p. 870)**, FILL).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

GMATRIX, *SYMFAC*, *Condname*, *NUMCOND*, --, *Matrixname*

Performs electric field solutions and calculates the self and mutual conductance between multiple conductors.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SYMFAC

Geometric symmetry factor. Conductance values are scaled by this factor which represents the fraction of the total device modeled. Defaults to 1.

Condname

Alphanumeric prefix identifier used in defining named conductor components.

NUMCOND

Total number of components. If a ground is modeled, it is to be included as a component.

--

Unused field.

Matrixname

Array name for computed conductance matrix. Defaults to GMATRIX.

Notes

To invoke the **GMATRIX (p. 862)** macro, the exterior nodes of each conductor must be grouped into individual components using the **CM (p. 356)** command. Each set of independent components is assigned a component name with a common prefix followed by the conductor number. A conductor system with a ground must also include the ground nodes as a component. The ground component is numbered last in the component name sequence.

A ground conductance matrix relates current to a voltage vector. A ground matrix cannot be applied to a circuit modeler. The lumped conductance matrix is a combination of lumped "arrangements" of voltage differences between conductors. Use the lumped conductance terms in a circuit modeler to represent conductances between conductors.

Enclose all name-strings in single quotes in the **GMATRIX (p. 862)** command line.

GMATRIX (p. 862) works with the following elements:

- SOLID5 (KEYOPT(1) = 9)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- SOLID98 (KEYOPT(1) = 9)
- LINK68
- PLANE230
- SOLID231
- SOLID232

This command is available from the menu path shown below only if existing results are available.

This command does not support multiframe restarts

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Component Based>Self/Mutual conductance

---

GMFACE, *Lab*, *N*

Specifies the facet representation used to form solid models.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid Labels:

FINE

Value that determines how coarse the facets will be.

N

An integer value between one (small) and ten (large) that determines the tolerances that will be applied to the creation of arcs and surfaces. Ten will create many facets, which may in turn cause Mechanical APDL to run very slowly. One will create fewer facets, which may in turn cause larger tolerance errors.

Menu Paths

Utility Menu>PlotCtrls>Style>Solid Model Facets

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***GO**, *Base*

Causes a specified line on the input file to be read next.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Base

"Go to" action:

:*label* --

A user-defined label (beginning with a colon (:), 8 characters maximum). The command reader will skip (and wrap to the beginning of the file, if necessary) to the first line that begins with the matching :*label*.

Caution:

This label option may not be mixed with do-loop or if-then-else constructs.

STOP --

This action will cause an exit from the Mechanical APDL program at this line.

Command Default

Read lines sequentially.

Notes

Causes the next read to be from a specified line on the input file. Lines may be skipped or reread. The ***GO (p. 864)** command will *not* be executed unless it is part of a macro, user file (processed by ***USE (p. 2059)**), an alternate input file (processed by **/INPUT (p. 948)**), or unless it is used in a batch-mode input stream. Jumping into, out of, or within a do-loop or an if-then-else construct to a :*label* line is not allowed.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/GO

Reactivates suppressed printout.

SESSION (p. 11): List Controls (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Reactivates printout suppressed with the **/NOPR (p. 1251)** command without producing any output. The **/GOPR (p. 865)** command has the same function except that it also produces a command response from the program.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

/GOLIST

Reactivates the suppressed data input listing.

SESSION (p. 11): List Controls (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Reactivates printout of the data input listing suppressed with **/NOLIST (p. 1249)**.

This command is valid in any processor, but only within a batch run (**/BATCH (p. 217)**).

Menu Paths

This command cannot be accessed from a menu.

/GOPR

Reactivates suppressed printout.

SESSION (p. 11): List Controls (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Reactivates printout suppressed with the **/NOPR (p. 1251)** command. The **/GO (p. 865)** command has the same function except that it does not produce a command response from the program.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

GPLOT

Controls general plotting.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command displays all entity types as specified via the **/GTYPE (p. 883)** command. Only selected entities (**NSEL (p. 1266)**, **ESEL (p. 661)**, **KSEL (p. 995)**, **LSEL (p. 1074)**, **ASEL (p. 192)**, **VSEL (p. 2163)**) will be displayed. See the descriptions of the **/GTYPE (p. 883)** and **/GCMD (p. 795)** commands for methods of setting the entity types displayed.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Dynamic Gap Cond>Define

Main Menu>Solution>Dynamic Gap Cond>Define

Utility Menu>Plot>Multi-Plots

/GRAPHICS, *Key*

Defines the type of graphics display.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Graphics key:

FULL

Display all model geometry and results.

POWER

Activate PowerGraphics (default when GUI is on).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

PowerGraphics ON (*Key* = POWER).

Notes

The **/GRAPHICS (p. 866)** command specifies the type of graphics display. *Key* = POWER activates the PowerGraphics capability. PowerGraphics offers faster plotting than the *Key* = FULL option, and speeds up element, results, area, line, and volume displays.

The default PowerGraphics mode is enabled automatically when accessing the GUI. This action supersedes all prior macros or start-up routines (`start.ans`, `config.ans`, etc.). Full graphics mode is accessed only by issuing **/GRAPHICS (p. 866)**,FULL *after* the GUI is active.

Results values (both printed and plotted) may differ between the *Key* = FULL and *Key* = POWER options because each option specifies a different set of data for averaging and display. For *Key* = FULL, all element and results values (interior and surface) are included. For *Key* = POWER, only element and results values along the model exterior surface are processed. Also, the element display can be different between the *Key* = FULL and *Key* = POWER options. For more information, see **EPLOT (p. 648)**.

When *Key* = FULL, it is possible to deselect an individual node, select all elements (including the element that contains that node), and then perform postprocessing calculations on those elements and have that unselected node not be considered in those calculations. If PowerGraphics is active, however, postprocessing always displays based on selected elements.

If you have specified one facet per element edge for PowerGraphics displays (via the **/EFACET (p. 602)** command or options from the **General Postproc** or **Utility** menu), PowerGraphics *does not plot midside nodes*. (**/EFACET (p. 602)** applies to element type displays only.)

Maximum values shown in plots can differ from printed maximum values. This is due to different averaging schemes used for plotted and printed maximum values.

When using solution coordinate systems for results output (**RSYS (p. 1639)**,SOLU) with PowerGraphics, the deformed or displaced shape in a POST1 contour display can be unexpected (although the contours are displayed in the expected colors). The program *does not* rotate displacement values (Ux,Uy,Uz) to global; instead, the displacements (stored locally) are added directly to the global coordinates (X,Y,Z). For example, if in PREP7 the nodes are rotated 90 degrees about the z axis and the global Uy displacements are relatively large, the Ux values will be large, causing the model to display a large deformation in the global X direction.

PowerGraphics displays do not average at geometric discontinuities. The printouts in PowerGraphics will, however, provide averaging information at geometric discontinuities if the models do not contain shell elements. Carefully inspect the data you obtain at geometric discontinuities.

PowerGraphics does not support the following diffusion analysis results: CONC, CG, DF, EPDI.

Key = FULL is not available for XFEM-based crack-growth analysis.

Excepting a few options, PowerGraphics supports these commands:

/CTYPE (p. 443),**PNUM (p. 1457)**
/EDGE (p. 594),**PSYMB (p. 1552)**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/ESHAPE (p. 665) **/SHELL (p. 1784)**
***GET (p. 797)** ***VGET (p. 2118)**

The following commands are executed via the *Key* = FULL option, regardless of whether PowerGraphics is activated:

/CTYPE (p. 443) **/ESYS (p. 685)** **/PBF (p. 1360)** **PRVECT (p. 1524)** ***VGET (p. 2118)**
DNSOL (p. 564) ***GET (p. 797)** **PLETAB (p. 1417)** **/PSYMB (p. 1552)** ***VPUT (p. 2149)**
/EDGE (p. 594) **SHELL (p. 1784)** **PLLS (p. 1419)** **SALLOW (p. 1644)**
ERNORM (p. 659) **/NORMAL (p. 1253)** **PRSECT (p. 1440)** **/SSCALE (p. 1847)**
ESORT (p. 680) **NSEL (p. 1266)** **PRETAB (p. 1496)** **/SHRINK (p. 1794)**
NSORT (p. 1283) **/PNUM (p. 1457)** **PRSECT (p. 1520)** **/TALLOW (p. 1897)**

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

/GRESUME, *Fname*, *Ext*, - -

Sets graphics settings to the settings on a file.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

The extension defaults to GSAV if *Fname* is blank.

- -

Unused field.

Notes

Causes a file to be read to reset the graphics slash (/) commands as they were at the last **/GSAVE (p. 875)** command.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Restore Plot Ctrls

/GRID, *KEY*

Selects the type of grid on graph displays.

GRAPHICS (p. 16): Graphs (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Grid key:

0 (OFF)

No grid.

1 (ON)

Full grid (X and Y grid lines).

2 (X)

Partial grid (X grid lines only).

3 (Y)

Partial grid (Y grid lines only)

Command Default

No grid.

Notes

Selects the type of grid on graph displays. Graphs with multiple Y-axes can have multiple grids (**/GRTYP (p. 874)**). The grid of the first curve is also used as the background grid (above and behind the curve). Grids for other curves are limited to be under the curves. See also **/GTHK (p. 882)** and **/GROPT (p. 870)** for other grid options.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/GROPT, *Lab*, *KEY*

Sets various line graph display options.

GRAPHICS (p. 16): Graphs (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Apply display style as selected from the following labels:

AXDV

Axis division (tick) marks (defaults to *KEY* = ON).

AXNM

Axis scale numbers (defaults to *KEY* = ON, which puts numbers at the back plane of the graph). If *KEY* = FRONT, numbers are on the front plane of the graph.

AXNSC

Axis number size scale factor. Input the scale value for *KEY* (defaults to 1.0).

ASCAL

Automatic scaling of additional Y-axes for multi-curve (**/GRTYP (p. 874)**, 2 or 3) graphs (defaults to *KEY* = ON). If *KEY* = OFF, use base Y-axis scaling (see the **/YRANGE (p. 2201)** command).

LOGX

Log X scale (defaults to *KEY* = OFF (linear)).

LOGY

Log Y scale (applies only to the base Y axis) (defaults to *KEY* = OFF (linear)).

FILL

Color fill areas under curves (defaults to *KEY* = OFF).

CGRID

Superimpose background grid (**/GRID (p. 869)**) over areas under filled curves (defaults to *KEY* = OFF).

DIG1

Number of significant digits before decimal point for axis values. Input the value for *KEY* (defaults to 4).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DIG2

Number of significant digits after decimal point for axis values. Input the value for *KEY* (defaults to 3).

VIEW

View key for graph displays (defaults to *KEY* = OFF, in which case the view is (0,0,1) for 2D graph displays or (1,2,3) for 3D graph displays). If *KEY* = ON, the view settings for graph displays are the same as the view settings for the model.

REVX

Plots the values on the X-axis in reverse order.

REVY

Plots the values on the Y-axis in reverse order.

DIVX

Determines the number of divisions (grid markers) that will be plotted on the X axis.

DIVY

Determines the number of divisions (grid markers) that will be plotted on the Y axis.

LTYP

Specifies whether program-generated (*KEY* = 1) or system-derived (*KEY* = 0) fonts are used for the axis labels.

CURL

Determines the position of the curve labels. If (*KEY* = 1), the curve label will be plotted in the legend column, and the label will be displayed in the same color as the curve. If (*KEY* = 0) the curve labels will be plotted near the curve. (default).

XAXO

When you use this label, the subsequent *KEY* value will determine an offset amount from the default (along the bottom) location for the X axis. If *KEY* = 1.0, a full offset occurs (the X axis is moved to the top of the graph). If *KEY* = 0.5, the axis is offset to the midpoint of the graph, and if *KEY* = 0 the axis remains in the original position, along the bottom of the graph. For any offset, a grey copy of the original axis (containing the axis numbering) remains at the original location.

YAXO

When you use this label, the subsequent *KEY* value will determine an offset amount from the default (along the left side of the graph) location for the Y axis. If *KEY* = 1.0, a full offset occurs (the Y axis is moved to the right side of the graph). If *KEY* = 0.5, the axis is offset to the midpoint of the graph, and if *KEY* = 0 the axis remains in the original position, along

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the left side of the graph. For any offset, a gray copy of the original axis (containing the axis numbering) remains at the original location.

KEY

Option values:

OFF (0)

Do not apply selected style.

ON (1)

Apply selected style.

nnnn

If *Lab* is DIG1 or DIG2, input the number of digits.

nn

If *Lab* is AXNSC, input the scale factor.

FRONT

If *Lab* is AXNM, FRONT may also be input.

Ndiv

If *Lab* is DIVX or DIVY, determines the number of divisions (1-99) that will be applied to the axis.

Kfont

If *Lab* is LTYP, *Kfont* is ON (1) or OFF(0). ON uses program-generated fonts for the axis labels, while OFF uses SYSTEM (Windows, X-system, etc.) fonts. Default: *Kfont* = ON (Mechanical APDL fonts).

Notes

Sets various line graph display options. Issue **/GROPT (p. 870)**,STAT to display the current settings.

Issue **/GROPT (p. 870)**,DEFA to reset default specifications.

Unless you issue **/GROPT (p. 870)**,VIEW,ON, the program indicates that graph-view manipulation is inactive.

See **/AXLAB (p. 216)**, **/GRTYP (p. 874)**, **/GRID (p. 869)**, and **/GTHK (p. 882)** for other graph-control options.

Automatic scaling using the **/XRANGE (p. 2199)** and **/YRANGE (p. 2201)** commands often yields inappropriate range values for logarithmic scales (**/GROPT (p. 870)**, LOGX or **/GROPT (p. 870)**, LOGY).

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

Utility Menu>PlotCtrls>Style>Graphs>Modify Curve

Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

GRP,*SIGNIF*,*Label*,--,*ForceType*

Specifies the grouping mode combination method.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single-point, multi-point, or DDAM response spectrum (*Spectype* = SPRS, MPRS, or DDAM on **SPOPT (p. 1837)**), the significance level of a mode is defined as the mode coefficient divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT (p. 1837)**,PSD.)

Label

Label identifying the combined mode solution output.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

--

Unused field.

ForceType

Label identifying the forces to be combined:

STATIC

Combine the modal static forces (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TOTAL

Combine the modal static plus inertial forces.

Notes

The *SIGNIF* value set with this command (including the default value of 0.001) overrides the *SIGNIF* value set with the **MXPAND (p. 1203)** command.

This command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine>GRP Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine>GRP Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine>GRP Method

Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine>GRP Method

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine>GRP Method

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine>GRP Method

---

/GRTYP, *KAXIS*

Selects single or multiple Y-axes graph displays.

GRAPHICS (p. 16): Graphs (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KAXIS

Axis selection key:

0 or 1

Single Y-axis. Up to 10 curves scaled to a single Y-axis.

2

Additional Y-axes (one for each curve) (3 curves maximum). Allows better scaling of curves with widely differing numbering ranges.

3

Same as 2 but with additional Y-axis and curves projected out of the plane (6 curves maximum). Allows clearer display with an isometric view. The default view when *KAXIS* = 3 is View,1,1,2,3.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Single Y-axis graph (except as noted for selection key 3).

Notes

The basic line graph has one or more curves plotted against the same Y and X axes. Multiple curve graphs can also be plotted with individual Y axes and the same X axis. The Y axis of the first curve is referred to as the base Y-axis and the Y axes of the other curves as additional Y axes. Curves are numbered sequentially from 1 (the base curve) in the order in which they are displayed. See the **/AXLAB (p. 216)**, **/GROPT (p. 870)**, **/GRID (p. 869)**, and **/GTHK (p. 882)** commands for other display options.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

/GSAVE, *Fname*, *Ext*, --

Saves graphics settings to a file for later use.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to GSAV if *Fname* is blank.

--

Unused field.

Notes

This command does not save all graphics settings, but only those that may be reset by the **/RESET (p. 1599)** command. The database remains untouched. Use the **/GRESUME (p. 868)** command to read the file. Repeated use of the **/GSAVE (p. 875)** command overwrites the previous data on the file. The following commands are saved by **/GSAVE (p. 875)**:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/ANGLE (p. 140) /DSCALE (p. 575) /GRTYP (p. 874) /PSF (p. 1537) /VIEW (p. 2131)
/AXLAB (p. 216) /EDGE (p. 594) /GTHK (p. 882) /PSYMB (p. 1552) /VSCALE (p. 2160)
/CLABEL (p. 349) /EFACET (p. 602) /GTYPE (p. 883) /RATIO (p. 1572) /VUP (p. 2173)
/COLOR (p. 400) /ESHAPE (p. 665) /LIGHT (p. 1049) /SHRINK (p. 1794) /WINDOW (p. 2177)
/CONTOUR (p. 434) /FOCUS (p. 771) /NORMAL (p. 1255) /SSCALE (p. 1840) /XRANGE (p. 2199)
/CPLANE (p. 426) /GCMD (p. 795) /NUMBER (p. 1269) /TRIAD (p. 2014) /YRANGE (p. 2201)
/CTYPE (p. 443) /GLINE (p. 860) /PBC (p. 1356) /TRLCY (p. 2015)
/CVAL (p. 450) /GRID (p. 869) /PLOPTS (p. 1435) /TYPE (p. 2034)
/DIST (p. 536) /GROPT (p. 870) /PNUM (p. 1457) /VCONE (p. 2078)

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Save Plot Ctrls

GSBDATA, *LabZ*, *VALUEZ*, *LabX*, *VALUEX*, *LabY*, *VALUEY*

Specifies the constraints or applies the load at the ending point for generalized plane strain option.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabZ

Constraint or load at the ending point in the fiber Z direction.

F

Apply a force in the fiber direction (default).

LFIBER

Define a length change in the fiber direction.

VALUEZ

Value for *LabZ*. The default is zero.

LabX

Constraint or load on rotation about X.

MX

Supply a moment to cause the rotation of the ending plane about X (default).

ROTX

Define a rotation angle (in radians) of the ending plane about X.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VALUEX

Value for *LabX*. The default is zero.

LabY

Constraint or load on rotation about Y

MY

Supply a moment to cause the rotation of the ending plane about Y (default).

ROTY

Define a rotation angle (in radians) of the ending plane about Y.

VALUEY

Value for *LabY*. The default is zero.

Notes

All inputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of Current-Technology Solid Elements in the *Element Reference*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Gen Plane Strain

Main Menu>Solution>Define Loads>Apply>Structural>Gen Plane Strain

---

GSGDATA, *LFIBER*, *XREF*, *YREF*, *ROTX0*, *ROTY0*

Specifies the reference point and defines the geometry in the fiber direction for the generalized plane strain element option.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LFIBER

Fiber length from the reference point. Defaults to 1.

XREF

X coordinate of the reference point. Defaults to zero.

YREF

Y coordinate of the reference point. Defaults to zero.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ROTX0

Rotation of the ending plane about X in radians Defaults to zero.

ROTY0

Rotation of the ending plane about Y in radians Defaults to zero.

Notes

The ending point is automatically determined from the starting (reference) point and the geometry inputs. All inputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of Current-Technology Solid Elements in the *Element Reference*.

Menu Paths

Main Menu>Preprocessor>Modeling>Genl plane strn

GSLIST, *Lab*

When using generalized plane strain, lists the input data or solutions.

SOLUTION (p. 38): FE Constraints (p. 47)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Specify the content to be listed.

GEOMETRY

List the data input using GSGDATA

BC

List the data input using GSBDATA.

REACTIONS

When the command is issued in POST1, list the reaction force at the ending point,

and the moment about X and Y if the corresponding constraints were applied.

RESULTS

When the command is issued in POST1, list the change of fiber length at the ending point during deformation and the rotation of the ending plane about X and Y during deformation.

ALL

List all of the above (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command can be used to list the initial position of the ending plane, the applied load or displacements in the fiber direction, the resulting position of the ending plane after deformation, and the available reaction forces and moments at the ending point.

All inputs and outputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of Current-Technology Solid Elements in the *Element Reference*.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Genl Plane Strn

GSSOL, *NVAR*, *Item*, *Comp*, *Name*

Specifies which results to store from the results file when using generalized plane strain.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR (p. 1302)**) while the name can be an eight byte character string. Overwrites any existing results for this variable.

Item

Label identifying item to be stored.

LENGTH

Change of fiber length at the ending point.

ROT

Rotation of the ending plane during deformation.

F

Reaction force at the ending point in the fiber direction.

M

Reaction moment applied on the ending plane.

Comp

Component of the item, if Item = ROT or M.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X

The rotation angle or reaction moment of the ending plane about X.

Y

The rotation angle or reaction moment of the ending plane about Y.

Name

Thirty-two character name identifying the item on the printout and display. Defaults to the label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

This command stores the results (new position of the ending plane after deformation) for generalized plane strain. All outputs are in the global Cartesian coordinate system. For more information about the generalized plane strain feature, see Generalized Plane Strain Option of Current-Technology Solid Elements in the *Element Reference*.

Menu Paths

Main Menu>TimeHist Postpro>Variable Viewer

/**GST**,*Lab*,*RunTrack*

Enables the Graphical Solution Tracking (GST) feature.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Enables or disables the GST feature:

ON – Enable

OFF – Disable

RunTrack

Enables or disables `.gst` file viewing in the Results Tracker utility:

ON – Enable

OFF – Disable (default)

Command Default

GST is enabled (*Lab* = ON) for interactive sessions and disabled (*Lab* = OFF) for batch runs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

For interactive sessions using the GUI (**/MENU (p. 1121)**,ON), GST directs solution graphics to the screen.

For interactive sessions not using the GUI (**/MENU (p. 1121)**,OFF), or for batch sessions, GST saves solution graphics to the `Jobname.gst` file. To create a `Jobname.gst` file that is compatible with the Results Tracker utility (available via the Mechanical APDL Product Launcher), issue **/GST (p. 880)**,ON,ON.

You can use the GST feature for these nonlinear analysis types: structural, thermal, electric, magnetic, fluid, and diffusion.

For more information about GST and illustrations of the GST graphics for each analysis type, see the analysis guide for the appropriate discipline.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Grph Solu Track

Main Menu>Solution>Load Step Opts>Output Ctrls>Grph Solu Track

---

GSUM

Calculates and prints geometry items.

PREP7 (p. 22): Keypoints (p. 25)
PREP7 (p. 22): Lines (p. 26)
PREP7 (p. 22): Areas (p. 27)
PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Calculates and prints geometry items (centroid location, moments of inertia, length, area, volume etc.) associated with the selected keypoints, lines, areas, and volumes. Geometry items are reported in the global Cartesian coordinate system. For volumes, a unit density is assumed unless the volumes have a material association via the **VATT (p. 2075)** command. For areas, a unit density (and thickness) is assumed unless the areas have a material (and real constant) association via the **AATT (p. 85)** command. For lines and keypoints, a unit density is assumed, irrespective of any material associations (**LATT (p. 1014)**, **KATT (p. 969)**, **MAT (p. 1111)**). Items calculated by **GSUM (p. 881)** and later retrieved by a ***GET (p. 797)** or ***VGET (p. 2118)** commands are valid only if the model is not modified after the **GSUM (p. 881)** command is issued. This command combines the functions of the **KSUM (p. 999)**, **LSUM (p. 1086)**, **ASUM (p. 203)**, and **VSUM (p. 2167)** commands.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Geometry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/GTHK, *Label*, *THICK*

Sets line thicknesses for graph lines.

GRAPHICS (p. 16): Graphs (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Apply thicknesses as selected from the following labels:

AXIS

Modify thickness of ordinate and abscissa axes on graph displays.

GRID

Modify thickness of grid lines on graph displays.

CURVE

Modify thickness of curve lines (when no area fill (**/GROPT (p. 870)**)).

THICK

Thickness ratio (whole numbers only, from -1 to 10):

-1

Do not draw the curve, but show only the markers specified by **/GMARKER (p. 861)**.

0 or 1

Thin lines.

2

The default thickness.

3

1.5 times the default thickness.

etc.

(up to 10)

Notes

Sets line thicknesses for graph lines (in raster mode only). Use **/GTHK (p. 882)**,STAT to show settings.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes
Utility Menu>PlotCtrls>Style>Graphs>Modify Curve
Utility Menu>PlotCtrls>Style>Graphs>Modify Grid

/GTYPE, *WN*, *LABEL*, *KEY*

Controls the entities that the GPLOT (p. 866) command displays.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which this command applies (defaults to 1)

LABEL

This represents the type of entity to display:

NODE

Nodes

ELEM

Elements

KEYP

Keypoints

LINE

Lines

AREA

Areas

VOLU

Volumes

GRPH

Graph displays

KEY

Switch:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Turns the entity type off.

1

Turns the entity type on.

Notes

The **/GTYPE (p. 883)** command controls which entities the **GPLOT (p. 866)** command displays. NODE, ELEM, KEYP, LINE, AREA, and VOLU are on by default. When ELEM is activated, you can control the type of element displayed via the **/GCMD (p. 795)** command (which also controls the type of graph display). When the GRPH entity type is activated, all other entity types are deactivated. Conversely, when any of the NODE, ELEM, KEYP, LINE, AREA, and VOLU entity types are active, the GRPH entity type is deactivated.

The **/GTYPE (p. 883)** command gives you several options for multi-window layout:

- One window
- Two windows (left and right or top and bottom of the screen)
- Three windows (two at the top and one at the bottom of the screen, or one top and two bottom windows
- Four windows (two at the top and two at the bottom)

Once you choose a window layout, you can choose one of the following: multiple plots, replotting, or no redisplay.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Multi-Plot Contrls

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

H Commands

HARFRQ, *FREQB*, *FREQE*, --, *LogOpt*, *FREQARR*, *Toler*

Defines the frequency range in a harmonic analysis.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FREQB

Frequency (Hz) at the beginning of the *FREQB* to *FREQE* range (if *FREQE* > *FREQB*). If *FREQE* is blank, the solution is done only at frequency *FREQB* (the central frequency of octave bands, when *LogOpt* = OB1, OB2, OB3, OB6, OB12 or OB24).

FREQE

Frequency at end of this range. For non-logarithm spacing (*LogOpt* is blank), solutions are done at an interval of (*FREQE* - *FREQB*) /*NSBSTP*, ending at *FREQE*, and no solution is done at the beginning of the frequency range, *FREQB*. *NSBSTP* is input via the **NSUBST (p. 1287)** command. See the **EXPSOL (p. 721)** command documentation for expansion pass solutions.

--

Reserved.

LogOpt

Logarithm frequency span. Solutions are done at an interval of (log(*FREQE*) - log(*FREQB*)) / (*NSBSTP*-1), (*NSBSTP*>1). The central frequency or beginning frequency is used for *NSBSTP* = 1. Valid values are:

OB1

Octave band.

OB2

1/2 octave band.

OB3

1/3 octave band.

OB6

1/6 octave band.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OB12

1/12 octave band.

OB24

1/24 octave band.

LOG

General logarithm frequency span.

FREQARR

An array containing frequency values (Hz). Combined with the tolerance argument, *Toler*, these values are merged with values calculated based on the specifications from *FREQB*, *FREQE*, and *LogOpt*, as well *NSBSTP* on the **NSUBST (p. 1287)** command and *Clust* on the **HROUT (p. 918)** command. Enclose the array name in percent (%) signs (for example, **HARFRQ (p. 885)**,,,,,%arrname%). Use ***DIM (p. 530)** to define the array.

Toler

Tolerance to determine if a user input frequency value in *FREQARR* is a duplicate and can be ignored. Two frequency values are considered duplicates if their difference is less than the frequency range multiplied by the tolerance. The default value is 1×10^{-5}.

Notes

Defines the frequency range for loads in the harmonic analysis (**ANTYPE (p. 162)**,HARMIC).

Do not use this command for a harmonic ocean wave analysis (**HROCEAN (p. 917)**).

When frequencies are user-defined, the array *FREQARR* must be one-dimensional and contain positive values. User-defined frequency input is not supported for the frequency-sweep Variational Technology method (**HROPT (p. 912)**,VT) or the harmonic balance method (**HROPT (p. 912)**,HBM).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substeps

Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substeps

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/HBC, *WN*, *Key*

Determines how boundary condition symbols are displayed in a display window.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window reference number. This number can be any window numbered 1 to 5, or ALL (for all active windows). Default = 1.

Key

Controls hidden-surface boundary condition display behavior:

ON, YES or 1 = Enable – Your boundary condition symbols are processed by the hidden-surface algorithm (for 2D graphics devices) and use an improved pressure-contour display (for 2D and 3D graphics devices).

OFF, NO or 0 (default) = Disable (default) – Your boundary condition symbols are *not* processed by the hidden-surface algorithm..

Command Default

/HBC (p. 887),*WN*,Off

For 2D graphics devices (such as X11, win32, and PNG), boundary condition symbols are not processed by the hidden-surface algorithm. All symbols are displayed.

For both 2D and 3D graphics devices, no enhanced rendering of pressure contours occurs.

Notes

With **/HBC (p. 887)**,*WN*,ON in effect, Mechanical APDL does *not* display symbols obscured by the model in the current view (that is, symbols inside of or behind the model are not drawn). This behavior lessens display clutter.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HBMAT,*Fname*,*Ext*,--,*Form*,*Matrx*,*Rhs*,*Mapping*

Writes an assembled global matrix in Harwell-Boeing format.

AUX2 (p. 61): Binary File Manipulation (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Defaults to the current Jobname if left blank.

Ext

Filename extension (eight-character maximum).

Defaults to `.matrix` if left blank.

--

Unused field.

Form

Specifies format of output matrix file:

ASCII

Write output matrix file in ASCII form.

BIN

Write output matrix file in binary form.

Matrx

Specify which matrix to write to the output matrix file:

STIFF

Write stiffness matrix to output matrix file. Valid for all types of analyses that write a `.full` file.

MASS

Write mass matrix to output matrix file. Valid for buckling, substructure, and modal analyses. If `.full` file was generated in a buckling analysis, then this label will write stress stiffening matrix to output matrix file.

DAMP

Write damping matrix to output matrix file. Only valid for damped modal analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rhs

Specifies whether to write the right-hand side vector to output matrix file:

YES

Write right-hand side vector to output matrix file.

NO

Do not write right-hand side vector to output matrix file.

Mapping

Specifies whether to write the mapping file. This file is always named *Fname*.mapping.

YES

Write the mapping file.

NO

Do not write the mapping file (default).

Command Default

By default, assuming a proper filename and extension have been entered, writes the stiffness matrix and right-hand side vector to Jobname.matrix in the current working directory in ASCII format.

Notes

This command copies a matrix from the assembled global matrix file (.full file) or from the superelement matrix file (.sub file) as specified on the **FILEAUX2 (p. 753)** command and write it in Harwell-Boeing format to a new file named Jobname.matrix. The Harwell-Boeing format is widely used by other applications that deal with matrices.

The assembled global matrix file is created during solution depending on the analysis type, equation solver, and other solution options. By default, the assembled global matrix file is never deleted at the end of solution. For most analysis types, the Sparse direct solver and the ICCG solver will write a .full file. All mode extraction methods used for buckling and modal analyses will write a properly formatted .full file to be used with the **HBMAT (p. 888)** command. However, when using distributed-memory parallel (DMP) processing, a majority of analyses will write a distributed (or local) form of the .full file which is not currently supported by the **HBMAT (p. 888)** command.

When dumping the stiffness matrix for transient and harmonic analyses, be aware that the element mass matrix values (and possibly element damping matrix values) are incorporated into the globally assembled stiffness matrix. Thus, the globally assembled stiffness matrix represents more than the stiffness of the model for these analysis types. Please refer to the *Mechanical APDL Theory Reference* for more details.

When dumping a .full file, the rows and columns corresponding to specified constraints (for example, **D (p. 483)** commands) are eliminated from the system of equations and therefore are not written to the .matrix file. Also, rows and columns corresponding to eliminated (dependent) degrees of freedom

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from coupling and/or constraint equations (for example, **CE (p. 297)**, **CP (p. 421)** commands) are also eliminated from the system of equations and are not written to the `.matrix` file. The DOFs that are eliminated from any coupling and/or constraint equations are determined internally by the solution code and may not match what you specified via the **CE (p. 297)**/**CP (p. 421)** (or similar) commands.

When dumping a `.sub` file, the full *n*x*n* matrix will be written to the `.matrix` file for either symmetric or unsymmetric matrices, regardless of whether any of the matrix coefficients are zero-valued. When dumping a `.full` file, only the lower triangular part of the matrix will be written to the `.matrix` file if the matrix is symmetric; the full matrix is written if the matrix is unsymmetric. Only matrix coefficients that are greater than zero will be written.

The Harwell-Boeing format is column-oriented. That is, non-zero matrix values are stored with their corresponding row indices in a sequence of columns. However, because the Mechanical APDL matrix files are stored by row and not column, when the **HBMAT (p. 888)** command is used with a non-symmetric matrix, the transpose of the matrix is, in fact, written.

The **WRFULL (p. 2184)** command, used with the **SOLVE (p. 1822)** command, generates the assembled global matrix file and eliminate the equation solution process and results output process.

The mapping file can be used to map the matrix equation numbers found in the `.matrix` file directly to the corresponding node numbers and degrees of freedom.

When dumping a CMS `.sub` file, the last rows/columns of the matrix are non-physical degrees of freedom added internally by the CMS process and cannot be mapped directly to a node number or particular degree of freedom.

Menu Paths

Utility Menu>File>List>Binary Files

Utility Menu>List>Files>Binary Files

HBMOPT, *Option*, *Value1*, *Value2*, *Value3*, *Value4*, *Value5*

Specifies solution options for a harmonic balance method (HBM) analysis.

SOLUTION (p. 38): Nonlinear Options (p. 40)
SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Harmonic balance method analysis option. There is no default. The semantic labels for *Value1*, *Value2*, *Value3*, *Value4*, and *Value5* as well as their descriptions and default/range values depend on the option setting. See the command specifications for each option for details. Possible *Option* settings are:

AFT

Specify alternate frequency-time (AFT) procedure options. This command is optional. See Command Specification for *Option* = AFT (p. 892)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NR

Specify Newton-Raphson controls. This command is optional. See Command Specification for *Option* = NR (p. 892).

TRTOL

Specify trust-region solver tolerances. This command is optional. See Command Specification for*Option* = TRTOL (p. 893).

CONTSET

Required settings for a numerical continuation solution. You must issue this command to properly set up an HBM analysis. See Command Specification for *Option* = CONTSET (p. 893).

CONTTERM

Specify termination criteria when continuation is on. This command is optional. See Command Specification for *Option* = CONTTERM (p. 895).

UINIT

Specify a solution guess at the beginning frequency of the solution. This command is optional. See Command Specification for *Option* = UINIT (p. 895).

LINEAR

Specify if nonlinearities are considered. This command is optional. See Command Specification for *Option* = LINEAR (p. 897).

SCAL

Linearly scale the solution vector to improve the convergence rate. See Command Specification for *Option* = SCAL (p. 898).

CONTLIST

List all continuation parameters. This command is optional. *Value1* .. *Value5* are unused fields.

LIST

List all continuation parameters (except continuation). This command is optional. *Value1* .. *Value5* are unused fields.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Option* = AFT

HBMOPT (p. 890), AFT, *NP*, *NT*, - -, - -, - -

Table 179: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = AFT

Value	Label and description	Default Value	Range
Value1	***NP*** **Maximum number of time periods** *NP* must be a positive integer. Note that the effective number of time periods may be smaller than *NP* as the AFT transient is stopped when periodic convergence is reached.	4	$NP > 0$
Value2	***NT*** **Number of time points per period** *NT* cannot be less than 2*NH+1, where NH is the number of harmonics[a]. The value of NT is automatically adjusted if this criterion is not met.	128	$16 \leq NT \leq 16384\ (= 2^{14})$
Value3, ... *Value5*	unused fields		

[a] Specify the number of harmonics NH by issuing **HROPT (p. 912)**,HBM,*NH*.

Command Specification for *Option* = NR

HBMOPT (p. 890), NR, *MAXITER*, *MAXINITITER*, - -, - -, - -

Table 180: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = NR

Value	Label and description	Default Value	Range
Value1	***MAXITER*** **Maximum number of iterations at a frequency point**	25 in most cases. 100 if continuation is turned off or natural continuation is selected by issuing **HBMOPT (p. 890)**,CONTSET with *CorrMeth* = OFF or NAT.	$MAXITER \geq 0$
Value2	***MAXINITITER*** **Maximum number of iterations at initial solution, irrespective of continuation settings**	500	$MAXINITITER \geq 0$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value	Label and description	Default Value	Range
Value3, … *Value5*	unused fields		

Command Specification for *Option* = TRTOL

HBMOPT (p. 890), TRTOL, ***UTOL***, ***RTOL***, ***GTOL***, **- -**, **- -**

Table 181: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = TRTOL

Value	Label and description	Default Value	Range
Value1	***UTOL*** **Tolerance on the solution step size at any given iteration**	1e-15	*UTOL* > 0
Value2	***RTOL*** **Tolerance on the function residual norm**	1e-6	*RTOL* > 0
Value3	***GTOL*** **Tolerance on the gradient norm**	1e-12	*GTOL* > 0
Value4, *Value5*	unused fields		

Command Specification for *Option* = CONTSET

HBMOPT (p. 890), CONTSET, ***CorrMeth***, ***DS***, ***DSMIN***, ***DSMAX***, **- -**

Table 182: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = CONTSET

Value	Label and description	
Value1	***CorrMeth*** **Corrector method** Specify the corrector type or alternative continuation method. The solver uses natural continuation for the first step for all continuation methods. (See notes (p. 900) for details.)	Possible *CorrMeth* settings are: **OFF** No continuation. Similar to natural continuation except there is no heuristic step adjustment, and the maximum number of allowed nonconverged steps is 1. **ARC** Arc-length corrector (default) **ORT** Orthogonal corrector

Value	Label and description	
		NAT Natural continuation. No continuation method is used.
Value2	***DS*** **Initial step size** Step size for the first substep. In the continuation process, the step size determines the size of the tangent prediction. Because it fully determines the predicted point, it also influences the correction, depending on the corrector method as listed below. From the predicted point, corrections occur: **OFF** or **NAT** - along the frequency direction **ARC** - along the sphere of radius *DS*. **ORT** - in the plane orthogonal to the tangent direction. Since the step size depends on the solution DOFs, it does not correspond to the size between two consecutive computed frequencies.	No default value **Note:** You must always specify a *DS* value except when *CorrMeth* = OFF. **Range:** $DSMIN \leq DS \leq DSMAX$
Value3	***DSMIN***[a] **Minimum step size** If the step size equals the minimum step size during solution and the nonlinear equation solver fails to converge, the solver will exit.	**Default:** *DS*/50 **Range:** $0 < DSMIN < DSMAX$
Value4	***DSMAX***[a] **Maximum step size**	**Default:** 5**DS* **Range:** $DSMAX > DSMIN$
Value5	unused field	

[a] During solution, if an internal step adjustment causes the step size to go either below *DSMIN* or above *DSMAX*, it is adjusted back to the relevant minimum (*DSMIN*) or maximum (*DSMAX*) allowed step size.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Option* = CONTTERM

HBMOPT (p. 890), CONTTERM, *MAXSTEPS*, *FREQMIN*, *FREQMAX*, - -, - -

Table 183: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = CONTTERM

Value	Label and description	Default Value	Range
Value1	***MAXSTEPS*** **Maximum number of successful continuation steps**[a] If exceeded, the solver exits.	10000	*MAXSTEPS* $\geq$ 0
Value2	***FREQMIN*** **Minimum frequency** During continuation, if the frequency is below this value, the solver exits.	MIN(*FREQB*, *FREQE*)[b]	*FREQMIN* $\leq$ MIN(*FREQB*, *FREQE*)[b]
Value3	***FREQMAX*** **Maximum frequency** During continuation, if the frequency exceeds this value, the solver exits.	MAX(*FREQB*, *FREQE*)[b]	*FREQMAX* $\geq$ MAX(*FREQB*, *FREQE*)[b]
Value4, *Value5*	unused fields		

[a] A continuation step is considered successful when the nonlinear equation solver converges and successfully produces solution points in two successive substeps. The number of substeps is equal to the number of successful continuation steps + 1.

[b] Set the beginning and ending frequency using the **HARFRQ (p. 885)** command. Note that the beginning frequency can be larger than the ending frequency if the solution branch needs to be evaluated with decreasing frequency.

Command Specification for *Option* = UINIT

HBMOPT (p. 890), UINIT, *U0Flag*, *U0Name*, *U0dof*, *U0BackName*, - -

Table 184: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = UINIT

Value	Label and description	
Value1	***U0Flag*** **Initial solution guess**	Possible *U0Flag* settings are: **0 or ZERO (default)** Use zero nodal solution as the initial guess.

Value	Label and description	
		1 or APDL Use a user-defined APDL array as the initial guess.
Value2	***U0name*** **Name of the APDL array for the initial solution guess** Optional. If *U0Flag* = 1 or APDL, this APDL array is used to specify the initial solution guess. It must be a 1D array (***DIM (p. 530)**) containing the multiharmonic solution guess. The array size is (2*NH+1)*N_{DOF} where NH is the number of harmonics specified by **HROPT (p. 912)**,HBM,*NH* and N_{DOF} is the number of degrees of freedom, which depends on *U0dof*.	No default value *U0name* is limited to 32 characters.
Value3	***U0dof*** **Degree of freedom numbering of the initial solution guess** Optional. If *U0Flag* = APDL, this indicates the degree of freedom ordering of the initial guess.	Possible *U0dof* settings are: **INT (default)** Internal Ansys DOFs. Length N_{DOF}= numdof*lenbac (See FULL File Format in the *Programmer's Reference*) **SOLU** HBM solution DOFs. Length N_{DOF} = number of multiharmonic equations/(2NH+1)
Value4	***U0BackName*** **APDL array name for the back nodal mapping vector corresponding to the initial guess** Optional. If *Uodof* = INT, the DOF ordering at the time of solution (see Degree of Freedom Ordering in the *Ansys Parametric Design Language Guide*) may be different from the intended DOF ordering corresponding to the initial solution guess (*U0Name*). To address this, *U0BackName* is used to specify a back nodal mapping vector corresponding to *U0Name*. It	No default value *U0BackName* is limited to 32 characters.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value	Label and description	
	must be a 1D array (***DIM (p. 530)**) with size equal to the number of Ansys internal nodes. *U0BackName* is used to reorder the initial solution guess to match the internal node ordering at the time of solution. If it is not provided and *U0dof* = *INT*, it is assumed that the initial solution guess corresponds to the internal DOF ordering at the time of solution. **Note:** When using **HBMOPT (p. 890)**,UINIT,APDL,*U0Name*,INT to specify the initial solution guess *U0Name* in internal DOFs, it is recommended that you provide a *U0BackName* vector to avoid any inconsistency between the array provided and the internal node ordering. For example usage, see Example 3: Two jointed beams with frictional contact interface.	
Value5	unused field	

Command Specification for *Option* = LINEAR

HBMOPT (p. 890), LINEAR, *AFTkey*, - -, - -, - -, - -

Table 185: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = LINEAR

Value	Label and description	
Value1	***AFTkey*** **Key to ignore nonlinearities (no AFT)**	Possible *AFTkey* settings are: **0 or OFF (default)** Do not ignore nonlinearities **1 or ON** Ignore nonlinearities (valid only in the first loadstep).
Value2, ... *Value5*	unused fields	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Option* = SCAL

HBMOPT (p. 890), SCAL, *Type*, *USCAL*, *FSCAL*, *DOF*, *Back*

Table 186: Semantic labels and definitions of *Value1*, ... *Value5* for *Option* = SCAL

Value	Label and description	
Value1	***Type*** **Scale the solution vector by the selected scaling factor type and specification on *USCAL*.** **Note:** See scaling equations in Equation Solution and example usage of HBMOPT,SCAL in Scaling the solution vector in the *Harmonic Balance Method Analysis Guide*.	Possible *Type* settings are: **0 or NONE (default)** No scaling.[a] **1 or APDL** Scaling factors contained in a user-defined APDL array whose name is specified by *USCAL* are used. **2 or VALU** Scalar value specified for *USCAL* is used.
Value2	***USCAL*** **Name of the user-defined APDL array (if *Type* = 1 or APDL) or scalar value (if *Type* = 2 or VALU) used to scale the solution vector to improve convergence behavior.**	*USCAL* setting depends on *Type* specification as follows: **For *Type* = 1 or APDL** *USCAL* is the name (up to 32 characters) of a previously defined APDL array that contains scaling factors. It must be a 1D array (***DIM (p. 530)**, ***VEC (p. 2085)**, etc.) with size equal to the size of the solution vector. **For *TYPE* = 2 or VALU** *USCAL* is a scalar value.
Value3	***FSCAL*** **Scaling value for the circular frequency (ω).** *FSCAL* is only valid for *Type* = 1 (or APDL) or 2 (or VALU)	**Default:** 1.0 **Range:***FSCAL* > 0.0

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value	**Label and description**	
Value4	***DoF***	Possible *DoF* settings are:
	Degree of freedom numbering of the user-defined scaling vector (*USCAL*)	**INT (default)**
	Optional. If *Type* = 1 or APDL, this indicates the degree of freedom ordering of the user-defined scaling vector.	Internal Ansys DOFs. Length N_{DOF}= numdof*lenbac (See FULL File Format in the *Programmer's Reference*)
		SOLU
		HBM solution DOFs. Length N_{DOF} = number of multiharmonic equations/(2NH+1)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value	Label and description	
Value3	***Back*** **APDL array name for the back nodal mapping vector corresponding to the user-defined scaling vector** Optional. *Back* is used to specify a back nodal mapping vector corresponding to the user-defined scaling vector, *UScal*. It must be a 1D array (***DIM (p. 530)**) with size equal to the number of Ansys internal nodes. *Back* is used to reorder the user-defined scaling vector to match the internal node ordering at the time of solution. If it is not provided and *DoF* = *INT*, it is assumed that the user-defined scaling vector corresponds to the internal DOF ordering at the time of solution. **Note:** When using **HBMOPT (p. 890)**,SCAL,APDL,*UScal*,*FSCAL*,INT to specify a user-defined scaling vector *UScal* in internal DOFs, it is recommended that you provide a *Back* vector to avoid any inconsistency between the array provided and the internal node ordering. For example usage, see Scaling the solution vector in the *Harmonic Balance Method Analysis Guide*.	No default value. *Back* is limited to 32 characters.

[a] *UScal* is not used if *Type* = 0 or NONE.

Notes

This command is used to specify solution options for a harmonic balance method analysis (see Harmonic Balance Method Analysis Guide).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There is no command default (p. 7). For an HBM analysis (*Method* = HBM on **HROPT (p. 912)**), you must issue at least one of the following:

- **HBMOPT (p. 890)**, CONTSET, OFF
- **HBMOPT (p. 890)**, CONTSET, *CorrMeth*, *DS* where *CorrMeth* is blank or any setting except OFF.

No continuation solver options

For **HBMOPT (p. 890)**, CONTSET, OFF:

- **NSUBST (p. 1287)** is used to specify the number of substeps and calculate the fixed frequency step. The first substep is at *FREQB* + (*FREQE* - *FREQB*)/*NSBSTP* as in other harmonic analyses, where *FREQB* and *FREQE* are the beginning and ending frequencies of a solution specified with the **HARFRQ (p. 885)** command.
- *FREQB* $\leq$ *FREQE* is enforced.
- Values prescribed for the minimum and maximum frequency (via **HBMOPT (p. 890)**,CONTTERM,,,,*FREQMIN*,*FREQMAX*) for the continuation simulation are ignored.

Alternating frequency-time options

The alternating frequency-time (AFT) procedure and derivation of relevant Jacobians for a general nonlinearity is described in Alternating Frequency-Time Procedure in the *Harmonic Balance Method Analysis Guide*.

Trust-region solver options

An iterative process is used to solve the HBM nonlinear equations using a trust-region nonlinear equation solver as detailed in Equation Solution in the *Harmonic Balance Method Analysis Guide*. All tolerances (*UTOL*, *RTOL*, and *GTOL* specified by **HBMOPT (p. 890)**,TRTOL) are also stopping criteria. The only required convergence criteria is the tolerance on the functional residual norm (**HBMOPT (p. 890)**,TRTOL,,*RTOL*). The other tolerances on the gradient norm and the step size are additional convergence criteria that indicate stronger convergence if satisfied.

Example Usage

Example 1: 1-DOF Duffing Oscillator in the *Harmonic Balance Method Analysis Guide*

Example 2: Two Cantilever Beams with Frictionless Gap Contact in the *Harmonic Balance Method Analysis Guide*

Example 3: Two jointed beams with frictional contact interface in the *Harmonic Balance Method Analysis Guide*

VM319: Nonlinear Harmonic Analysis of a Cyclic Chain of Oscillators

VM320: Nonlinear Harmonic Analysis of a Frictional Damper

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

/HEADER,*Header*,*Stitle*,*Idstmp*,*Notes*,*Colhed*,*Minmax*

Sets page and table heading print controls.

POST1 (p. 51): Listing (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Header

Mechanical APDL page header (system, date, time, version, copyright, title, etc.):

ON

Turns this item on (default for batch mode; not available for interactive mode).

OFF

Turns this item off.

(blank)

Retains the previous setting.

Stitle

Subtitles (see **/STITLE (p. 1867)** command): ON, OFF, or (blank) (see above).

Idstmp

Load step information (step number, substep number, time value): ON, OFF, or (blank) (see above).

Notes

Information relative to particular table listings: ON, OFF, or (blank) (see above).

Colhed

Column header labels of table listings (currently only for single column tables): ON, OFF, or (blank) (see above).

Minmax

Minimum/maximum information or totals after table listings: ON, OFF, or (blank) (see above).

Command Default

All specifications are on (batch mode); *Header* = OFF, all other specifications are on (interactive mode)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Sets specifications on or off for page and table heading print controls associated with the POST1 **PRNSOL (p. 1505)**, **PRESOL (p. 1481)**, **PRETAB (p. 1490)**, **PRRSOL (p. 1518)**, and **PRPATH (p. 1517)** commands.

Note:

If the printout caused a top-of-form (page eject to top of next page), the top-of-form is also suppressed with the printout. Issue **/HEADER (p. 902)**,STAT to display the current settings. Issue **/HEADER (p. 902)**,DEFA to reset the default specifications.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

HELP, *Name*

Displays help information for Mechanical APDL commands and element types.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Name

Command name or element type. Examples are: **HELP (p. 903)**,MP or **HELP (p. 903)**,SOLID185 (or **HELP (p. 903)**,185). For a list of elements of a particular type, enter **HELP (p. 903)**,BEAM, **HELP (p. 903)**,SOLID, **HELP (p. 903)**,HYPER, etc.

Notes

If *Name* matches a command or element name, the description for that command or element is displayed in the Help window.

For command help, type the complete command name (including the * or /). The help system cannot find partial matches. If * is used at the beginning of the string, it is interpreted as a Mechanical APDL * command.

For help on topics that are not Mechanical APDL commands or element types (for example, help for the word "material"), use the full text-search features of the Mechanical APDL help system.

The **HELP (p. 903)** command is valid only in GUI mode. To obtain help when not in GUI mode, activate the GUI (**/MENU (p. 1121)**,ON) or activate the help system directly (**/UI (p. 2040)**,HELP).

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

HEMIOPT, *HRES*, *TOLERANCE*

Specifies options for Hemicube view factor calculation.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

HRES

Hemicube resolution. Increase value to increase the accuracy of the view factor calculation. Defaults to 10.

TOLERANCE

Tolerance value that controls whether or not facets are subdivided in view factor calculations to increase view factor accuracy. *TOLERANCE* is closely related to the spacing between facets. Defaults to 1e-6.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>View Factor

Main Menu>Radiation Opt>Radiosity Meth>View Factor

Main Menu>Solution>Radiation Opts>View Factor

HFANG, *Lab*, *PHI1*, *PHI2*, *THETA1*, *THETA2*

Defines or displays spatial angles of a spherical radiation surface for sound radiation parameter calculations.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Spatial angle label.

ANGLE

Define spatial angles (default).

STATE

Display spatial angles. *PHI1*, *PHI2*, *THETA1*, and *THETA2* are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PHI1, PHI2

Starting and ending φ angles (degrees) in the spherical coordinate system. *PHI1* defaults to 0 and *PHI2* defaults to 360.

THETA1, THETA2

Starting and ending θ angles (degrees) in the spherical coordinate system. *THETA1* defaults to 0 and *THETA2* defaults to 180.

Notes

Defines or displays spatial angles of a spherical radiation surface.

Use this command only with **PLFAR (p. 1412)**,*Lab* = PRES, or **PRFAR (p. 1490)**,*Lab* = PRES.

Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain

Main Menu>General Postproc>List Results>Field Extension>Efficiency

Main Menu>General Postproc>List Results>Field Extension>Power Gain

Main Menu>General Postproc>List Results>Field Extension>Rad Power

Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain

---

HFSYM,*KCN*,*Xkey*,*Ykey*,*Zkey*

Indicates the presence of symmetry planes for the computation of acoustic fields in the near and far field domains (beyond the finite element region).

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Coordinate system reference number. *KCN* may be 0 (Cartesian), or any previously defined local Cartesian coordinate system number (>10). Defaults to 0.

Xkey

Key for acoustic field boundary condition, as prescribed for the solution, corresponding to the x = constant plane:

None

No sound soft or sound hard boundary conditions (default).

SSB

Sound soft boundary (pressure = 0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHB

Sound hard boundary (normal velocity = 0).

Ykey

Key for acoustic field boundary condition, as prescribed for the solution, corresponding to the y = constant plane:

None

No sound soft or sound hard boundary conditions (default).

SSB

Sound soft boundary (pressure = 0).

SHB

Sound hard boundary (normal velocity = 0).

Zkey

Key for acoustic field boundary condition, as prescribed for the solution, corresponding to the z = constant plane:

None

No sound soft or sound hard boundary conditions (default).

SSB

Sound soft boundary (pressure = 0).

SHB

Sound hard boundary (normal velocity = 0).

Notes

HFSYM (p. 905) uses the image principle to indicate symmetry planes (x, y, or z = constant plane) for acoustic field computations outside the modeled domain. A sound hard boundary condition must be indicated even though it occurs as a natural boundary condition.

No menu paths are available for acoustic applications.

Menu Paths

Main Menu>General Postproc>List Results>Field Extension>Direct Gain

Main Menu>General Postproc>List Results>Field Extension>Efficiency

Main Menu>General Postproc>List Results>Field Extension>Far Field

Main Menu>General Postproc>List Results>Field Extension>Near Field

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>General Postproc>List Results>Field Extension>Pattern
Main Menu>General Postproc>List Results>Field Extension>Power Gain
Main Menu>General Postproc>List Results>Field Extension>Rad Power
Main Menu>General Postproc>List Results>Field Extension>RCS
Main Menu>General Postproc>List Results>Field Extension>RCS Normalized
Main Menu>General Postproc>Path Operations>Map onto Path>HF Near Field
Main Menu>General Postproc>Plot Results>Field Extension>Direct Gain
Main Menu>General Postproc>Plot Results>Field Extension>Far Field
Main Menu>General Postproc>Plot Results>Field Extension>Pattern
Main Menu>General Postproc>Plot Results>Field Extension>RCS
Main Menu>General Postproc>Plot Results>Field Extension>RCS Normalized

---

*HPROD, *A*, *B*, *C*

Performs a Hadamard vector product (C = A B).

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

A

Name of vector A. Must have been previously created by a ***VEC (p. 2085)** command.

B

Name of vector B. Must have been previously created by a ***VEC (p. 2085)** command.

C

Name of vector C. Must be specified (no default).

Notes

For two vectors *A* and *B* of the same dimension *n*, the Hadamard product (A B) is a vector of the same dimension as the operands, with elements given by:

$$(A \circ B)_i = A_i * B_i$$

This command is limited to vector operands.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HPTCREATE, *TYPE*, *ENTITY*, *NHP*, *LABEL*, *VAL1*, *VAL2*, *VAL3*

Defines a hard point.

PREP7 (p. 22): Hard Points (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TYPE

Type of entity on which the hard point will be created.

LINE

Hard point will be created on a line.

AREA

Hard point will be created within an area (not on the boundaries).

ENTITY

Number of the line or area on which the hard point will be created.

NHP

Number assigned to the hard point. Defaults to the lowest available hard point number.

LABEL

If *LABEL* = COORD, *VAL1*, *VAL2*, and *VAL3* are the respective global X, Y, and Z coordinates. If *LABEL* = RATIO, *VAL*1 is the parameter value (this is available only for lines). Valid parameter values are between 0 and 1. *VAL2* and *VAL3* are ignored.

VAL1

If *LABEL* = RATIO, ratio value for line. If *LABEL* = COORD, global X coordinate value.

VAL2

If *LABEL* = COORD, global Y coordinate value.

VAL3

If *LABEL* = COORD, global Z coordinate value.

Notes

The ability to enter a parameter value provides a simple way of positioning hard points on lines. For example, to place a hard point halfway along a line, one can simply specify a *VAL1* value of 0.5.

For models imported through the DEFAULT IGES filter, you can place hard points on models only by specifying coordinates (you can't place a hard point using interactive picking).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you issue any commands that update the geometry of an entity, such as Boolean or simplification commands, any hard points associated with that entity are deleted. Therefore, you should add any hard points after completing the solid model. If you delete an entity that has associated hard points, those hard points are either

- Deleted along with the entity (if the hard point is not associated with any other entities).
- Detached from the deleted entity (if the hard point is associated with additional entities).

When archiving your model (**CDWRITE (p. 293)**), hardpoint information cannot be written to the IGES file. The `Jobname.cdb` file can be written with the **CDWRITE (p. 293)**,DB option.

Hard points are only applicable for area and volume meshing, not for beams.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on area>Hard PT by coordinates

Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on line>Hard PT by coordinates

Main Menu>Preprocessor>Modeling>Create>Hard Points>Hard PT on line>Hard PT by ratio

Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on area>Hard PT by coordinates

Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on area>Hard PT by picking

Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by coordinates

Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by picking

Main Menu>Preprocessor>Modeling>Create>Keypoints>Hard PT on line>Hard PT by ratio

---

HPTDELETE, *NP1*, *NP2*, *NINC*

Deletes selected hardpoints.

PREP7 (p. 22): Hard Points (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1*, *NP2*, *NINC

Delete the pattern of hard points beginning with *NP1* to *NP2* in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is all selected hard points (**KSEL (p. 995)**). If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored.

Notes

Deletes all attributes attached to the designated hard points as well as the hard points themselves. If any entity is attached to a designated hard point, the command detaches the hard point from that entity (the program will alert you that this will occur).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Hard Points

HRCPLX, *LOADSTEP*, *SUBSTEP*, *OMEGAT*, *1STLCASE*, *2NDLCASE*

Computes and stores in the database the time-harmonic solution at a prescribed phase angle.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LOADSTEP

Load step number of the data set to be read (defaults to 1).

SUBSTEP

Substep number within *LOADSTEP*.

OMEGAT

Angle in degrees (Ω (angle) times T (time)).

- If $\geq$ 360°, the amplitude is supplied.
- All others supply results at that angle. For example, if the angle is set to 0.0°, the real part of the solution is supplied. If the angle is set to -90° the imaginary part of the solution is supplied.

1STLCASE

First load case number (defaults to 1).

2NDLCASE

Second load case number (defaults to 2).

Notes

HRCPLX (p. 910) invokes a macro that combines the real and imaginary parts of the solution. If the angle is specified, it produces the following:

$$\{R\} = \{R_R\}\cos\alpha - \{R_I\}\sin\alpha$$

Where:

R_R and R_I are, respectively, the real and imaginary parts of the results quantity (e.g. the nodal displacements, the reaction forces, ...).

α is the angle (OMEGAT).

1STLCASE and *2NDLCASE* are internally generated load cases. You may want to specify these to avoid overwriting an existing load case number 1 or 2.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Not all results computed by this command are valid. See Summable, Non-Summable and Constant Data in the *Basic Analysis Guide* for more information. When the amplitude of the solution is requested (*OMEGAT* >= 360°), averaged values (such as the nodal component stresses, which are an average of element nodal component stresses) are calculated by averaging the amplitudes. Because the degrees of freedom results have different phases, derived results (such as the equivalent stress SEQV) are not valid. See POST1 and POST26 – Complex Results Postprocessing for more details about post-processing complex results.

For postprocessing amplitudes, the only appropriate coordinate system is the solution coordinate system (**RSYS (p. 1639)** ,SOLU). When displaying the displacement amplitudes, use a contour display (**PLNSOL (p. 1425)** command). Because a deformed shape display (**PLDISP (p. 1400)** command) could lead to a non-physical shape, the displacement scaling is off by default (**/DSCALE (p. 575)**,,OFF).

For postprocessing cylindrical geometry, it is suggested that you rotate the element coordinate systems into the appropriate cylindrical system (*EMODIF,,ESYS*) before running the solution and then view the results in this system (*RSYS,SOLU*) in POST1.

Since **HRCPLX (p. 910)** performs load case combinations, it alters most of the data in the database. In particular, it alters applied loads such as forces and imposed displacements. To restore the original loads in the database for a subsequent analysis, reissue the **SET (p. 1724)** command in POST1 to retrieve the real and imaginary set data.

To animate the solution over one period, use the **ANHARM (p. 141)** command.

OMEGAT is not equal to the phase shift.

This command is not supported after a cyclic symmetry analysis; use **/CYCEXPAND (p. 454)**,,PHASEANG instead.

Menu Paths

This command cannot be accessed from a menu.

HREXP,*ANGLE*

Specifies the phase angle for the harmonic analysis expansion pass.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ANGLE

Phase angle (degrees) for expansion pass. If ALL (default), use both 0.0° (real) and 90.0° (imaginary) phase angles.

Notes

Specifies the phase angle where the expansion pass will be done for a harmonic mode-superposition expansion pass.

For a specific angle, the following real solution is stored in the results (*.rst) file:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\{u\}=\{u^i_{\max}\cos(\phi^i-\varphi)\}$$

Where:

i is the degree of freedom number.

$u^i_{\max}$ is the amplitude of the i th degree of freedom solution

Φ^i is the phase shift angle of the i th degree of freedom solution

φ is the supplied phase shift angle (*ANGLE*)

If *ANGLE* is ALL, both the real and imaginary parts of the solution are stored in the results file.

For more details about the solution equations, see Harmonic Analyses in the *Mechanical APDL Theory Reference*.

This command is ignored if the **HROPT (p. 912)** command has been issued with *Method* = KRYLOV, VT, or VTPA.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Load Step

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Load Step

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>By Time/Freq

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's

---

HROPT, *Method*, *Value1*, *Value2*, *Value3*, *Value4*, *Value5*

Specifies harmonic analysis options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Method

Solution method for the harmonic analysis:

AUTO

Automatically select the most efficient method (default). Either the full method (FULL) or the frequency-sweep method (VT) is selected, depending on the model. For *Method* = AUTO, *Value1*..*Value5* are unused fields.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FULL

Full method. For *Method* = AUTO, *Value1*..*Value5* are unused fields.

MSUP

Mode-superposition method. See Command Specification for *Method* = MSUP (p. 913).

VT

Frequency-sweep (Variational Technology) method (based on the FULL harmonic algorithm). See Command Specification for *Method* = VT or VTPA (p. 914).

VTPA

Frequency-sweep (Variational Technology) perfect absorber method (based on the FULL harmonic algorithm). See Command Specification for *Method* = VT or VTPA (p. 914).

KRYLOV

Frequency-sweep Krylov method. See Command Specification for *Method* = KRYLOV (p. 915).

Note:

The Krylov approximation can be run using macros as customizable templates as described in Frequency-Sweep Harmonic Analysis via the Krylov Method in the *Structural Analysis Guide*. Alternatively, you can perform a Krylov solution without customization as described in Krylov Method Implemented using Mechanical APDL Commands in the *Structural Analysis Guide*.

HBM

Harmonic Balance method. See Command Specification for *Method* = HBM (p. 916).

If the solution method is not specified, the program automatically selects either the Full method or the frequency-sweep method, depending on which method is most efficient for the model. The frequency-sweep method uses the underlying Variational Technology method.

Command Specification for *Method* = MSUP

HROPT (p. 912), MSUP, *MAXMODE*, *MINMODE*, *MCFwrite*, *--*, *MCkey*

For more details related to mode-superposition harmonic analyses, see Notes (p. 916).

Table 187: Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = MSUP

Value	MSUP label and description
Value1	***MAXMODE*** Largest mode number to be used to calculate the response (for *Method* = MSUP only). Defaults to the highest mode calculated in the preceding modal analysis.

Value	MSUP label and description
Value2	***MINMODE*** Smallest mode number to be used (for *Method* = MSUP only). Defaults to 1.
Value3	***MCFwrite*** Modal coordinates output key to the Jobname.mcf file (valid only for the mode-superposition method). To control how Jobname.mcf is written, specify options on the **MCFOPT (p. 1116)** command. **NO** Modal coordinates are not written to Jobname.mcf. **YES** Modal coordinates are written to the text file Jobname.mcf (default). **Note:** *MCFwrite* provides a singular (one-time) output of the modal coordinates. It will not provide the information for **LSWRITE (p. 1087)**/**LSSOLVE (p. 1085)** operations.
Value4	**--** Unused field.
Value5	***MCkey*** Modal coordinates output key to the .rfrq file (valid only for the mode-superposition method): **AUTO** Writing depends on the modal analysis settings of the **MXPAND (p. 1203)** command (default). **YES** Always write the modal coordinates to the file Jobname.rfrq. A subsequent expansion pass (**EXPASS (p. 714)**) is not supported.

Command Specification for *Method* = VT or VTPA

HROPT (p. 912), **VT**, **--**, **--**, **--**, ***Damp***, **--**

or

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HROPT (p. 912), VTPA, - -, - -, - -, *Damp*, - -

Table 188: Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = VT or VTPA

Value	VT (or VTPA) label and description
Value1, *Value2*, *Value3*	**- -** Unused fields.
Value4	***Damp*** Damping mode for frequency-dependent material properties (valid only for the frequency-sweep method, *Method* = VT). **Hysteretic** Not proportional to the frequency. **Viscous** Proportional to the frequency (default).
Value5	**- -** Unused field.

Command Specification for *Method* = KRYLOV

HROPT (p. 912), KRYLOV, *FRQKRY*, - -, - -, - -, - -

Table 189: Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = KRYLOV

Value	KRYLOV label and description
Value1	***FRQKRY*** Frequency at which the Krylov subspace is built. Defaults to the middle of the frequency range specified using the **HARFRQ (p. 885)** command. (*FRQKRY* is not supported for a distributed-memory parallel solution that uses frequency domain decomposition, **DDOPTION (p. 508)**,FREQ).
Value2, ... *Value5*	**- -** Unused fields.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specification for *Method* = HBM

HROPT (p. 912), HBM, *NH*, - -, - -, - -, - -

Table 190: Semantic labels and definitions of *Value1*, ... *Value5* for *Method* = HBM

Value	HBM label and description
Value1	***NH*** Number of harmonics used in a harmonic balance method analysis. It must be an integer within the range $1 \leq NH \leq 30$. Defaults to 1.
Value2, ... *Value5*	**- -** Unused fields.

Command Default

If you run a harmonic analysis (*Antype* = HARMIC or 3 on the **ANTYPE (p. 162)** command) and you do not issue **HROPT (p. 912)**, the default value is *Method* = AUTO.

Notes

Specifies the method of solution for a harmonic analysis (**ANTYPE (p. 162)**,HARMIC). For detailed descriptions of the various solution methods, see Overview of Harmonic Analysis Solution Methods in the *Structural Analysis Guide*.

If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

For *Method* = MSUP:

For cyclic symmetry mode-superposition harmonic solutions, *MAXMODE* and *MINMODE* are ignored.

To include residual vectors in your mode-superposition harmonic analysis, specify **RESVEC (p. 1603)**,ON.

MAXMODE and *MINMODE* are ignored after a modal restart analysis where remote modal files usage (**MODDIR (p. 1134)**) and residual vector calculation (**RESVEC (p. 1603)**,ON) have been activated.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HROCEAN, *Type*, *NPHASE*

Perform the harmonic ocean wave procedure (HOWP).

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Specifies how to include ocean wave information in a harmonic analysis:

HARMONIC

Performs a harmonic analysis using both real and imaginary load vectors calculated via the harmonic ocean wave procedure (HOWP). This behavior is the default. This option performs a harmonic analysis running at a frequency determined by the wave period (specified via **OCTABLE (p. 1316)** command input).

STATIC

Performs a static analysis using both real and imaginary load vectors (calculated via HOWP). This option works by performing a harmonic analysis running at a frequency of 0.0.

OFF

Deactivates a previously activated HOWP and performs a *standard* harmonic analysis.

NPHASE

Positive number specifying the number of phases to calculate forces. This value must be at least 8. The default value is 20.

Command Default

The default **HROCEAN (p. 917)** command (issuing the command with no arguments) is:

HROCEAN (p. 917),HARMONIC,20

The command applies HOWP when ocean wave information is available (that is, when ocean wave information has been specified via the **OCDATA (p. 1307)** and **OCTABLE (p. 1316)** commands). If this command is issued when no ocean wave information is available, the program generates an error message.

Notes

The **HROCEAN (p. 917)** command applies ocean wave information (obtained via the **OCDATA (p. 1307)** and **OCTABLE (p. 1316)** commands) in a harmonic analysis (**ANTYPE (p. 162)**,HARMIC) as real and imaginary forces.

You can apply only one ocean load at a time.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The applied frequency in the harmonic (*Type* = HARMONIC) analysis is based on the wave period input on the **OCTABLE (p. 1316)** command (and not on **HARFRQ (p. 885)** command input, which cannot be used). Phase-shift input on the **OCTABLE (p. 1316)** command is ignored.

HOWP does not generate a damping matrix. If you require a damping matrix, you must add it separately.

The command applies to regular wave types only (Airy with one wave component, Wheeler with one wave component, Stokes, and stream function). Irregular wave types are not supported. For information about wave types, see Hydrodynamic Loads in the *Mechanical APDL Theory Reference*.

The program calculates the forces on each load component of each element at *NPHASE* solutions, spread evenly over one wave cycle. Then, the minimum and maximum, and the phase between them, are calculated. The command uses the resulting information to generate the real and imaginary loads.

HOWP cannot be used with stress stiffening.

HOWP works with the full harmonic analysis method (**HROPT (p. 912)**,FULL) only.

For more information, see Harmonic Ocean Wave Procedure (HOWP) in the *Mechanical APDL Theory Reference*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substeps

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>HOWP and Substeps

Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substeps

Main Menu>Solution>Load Step Opts>Time/Frequenc>HOWP and Substeps

---

HROUT,*Reimky*,*Clust*,*Mcont*,*EngCalc*

Specifies the harmonic analysis output options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Reimky

Real/Imaginary print key:

ON

Print complex displacements as real and imaginary components (default).

OFF

Print complex displacements as amplitude and phase angle (degrees).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clust

Cluster option (for **HROPT (p. 912)**,MSUP):

OFF

Uniform spacing of frequency solutions (default).

ON

Cluster frequency solutions about natural frequencies.

Mcont

Mode contributions key (for **HROPT (p. 912)**,MSUP):

OFF

No print of mode contributions at each frequency (default).

ON

Print mode contributions at each frequency.

EngCalc

Additional element energies calculation key:

NO

Do not calculate additional energies (default).

YES

Calculate average, amplitude, and peak values for the following: stiffness and kinetic energies, damping energy, and work done by external loads.

Notes

Specifies the harmonic analysis (**ANTYPE (p. 162)**,HARMIC) output options. If used in SOLUTION, this command is valid only within the first load step. **OUTPR (p. 1332)**,NSOL must be specified to print mode contributions at each frequency.

If the calculation of additional energies is requested (*EngCalc* = YES) in a mode-superposition harmonic analysis (*Method* = MSUP on the **HROPT (p. 912)** command), work done by external loads is not calculated in the expansion pass if *MSUPkey* = YES on the **MXPAND (p. 1203)** command. If *MSUPkey* = NO, work due to element loads is calculated in the expansion pass of the real part, but not work due to nodal loads. The average, amplitude, and peak potential energies for complex results (Potential Energy (or Stiffness Energy) for Complex Results) calculated for a linear perturbation full harmonic analysis with a nonlinear base analysis are approximate solutions and may not be accurate.

Only the *Reimky* argument is supported and applicable for frequency-sweep harmonic analyses using the Krylov or Variational Technology Methods. All other arguments are ignored if the **HROPT (p. 912)** command has been issued with *Method* = KRYLOV, VT, or VTPA.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I Commands

IC, *NODE*, *Lab*, *VALUE*, *VALUE2*, *NEND*, *NINC*, *LSIC*

Specifies initial conditions at nodes.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node at which initial condition is to be specified. If ALL, apply to all selected nodes (**NSEL (p. 1266)**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE*.

Lab

Degree-of-freedom label for which the initial condition is to be specified. If ALL, use all appropriate labels.

Structural labels: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations); HDSP (hydrostatic pressure); PRES (pore pressure)

For structural static and transient analyses, specify translational and rotational velocities as initial conditions using these labels: VELX, VELY, VELZ (translational velocities); OMGX, OMGY, OMGZ (rotational velocities).

For structural transient analysis, specify translational and rotational accelerations as initial conditions using these labels: ACCX, ACCY, ACCZ (translational accelerations); DMGX, DMGY, DMGZ (rotational accelerations).

The velocity and acceleration initial conditions are *not* included with *Lab* = ALL.

Thermal labels: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature)

Magnetic labels: MAG (scalar magnetic potential); AZ (vector magnetic potential)

Diffusion label: CONC (concentration)

Acoustic label: ENKE (acoustic energy density)

VALUE

Initial value of the degree of freedom. Defaults to the program default for that degree of freedom (0.0 for structural analysis, **TUNIF (p. 2027)** for thermal analysis, etc.). Values are in the nodal coordinate system and in radians for rotational degrees of freedom.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VALUE2

Second-order degree of freedom value, mainly used for non-structural DOF where VELX can't be used. Defaults to the program default for that degree of freedom (0.0 for structural analysis). Values are in the nodal coordinate system and in radians/time for rotational degrees of freedom.

NEND, NINC

Specifies the same initial condition values at the range of nodes from *NODE* to *NEND* (defaults to *NODE*), in steps of *NINC* (defaults to 1).

LSIC

Apply the **IC (p. 921)** command at this load step instead of at the first load step.

Notes

The **IC (p. 921)** command specifies initial conditions, which are the initial values of the specified degrees of freedom. It is valid only for a static analysis and full method transient analysis (**TIMINT (p. 2001)**,ON and **TRNOPT (p. 2017)**,FULL). For the transient, the initial value is specified at the beginning of the first load step, that is, at time = 0.0.

To specify initial conditions at a load step other than the first step, use the *LSIC* field. Specify a load step in the *LSIC* field to apply the **IC (p. 921)** command at that load step. The value of the variable calculated from the solver will be overwritten by the one being specified by the **IC (p. 921)** command at the start time for that load step.

If constraints (**D (p. 483)**, **DSYM (p. 580)**, etc.) and initial conditions are applied at the same node, the constraint specification overrides. Exercise caution when specifying constraints. The degree-of-freedom values start from zero, or the first value given in the table when table name is specified. To match the nonzero initial condition value with the initial value for degree-of-freedom constraint, use a table for the degree-of-freedom constraint.

For thermal analyses, any **TUNIF (p. 2027)** specification should be applied before the **IC (p. 921)** command. Otherwise, the **TUNIF (p. 2027)** specification is ignored. If the **IC (p. 921)** command is input before any **TUNIF (p. 2027)** specification, use the **ICDELE (p. 923)** command and then reissue any **TUNIF (p. 2027)** specification and then follow with the **IC (p. 921)** command.

When issuing the **IC (p. 921)** command for elements SOLID278 Layered Thermal Solid and SOLID279 Layered Thermal Solid with through-the-thickness degrees of freedom (KEYOPT(3) = 2), layers are always interpolated linearly based on the location of the degrees of freedom.

Define consistent initial conditions. For example, if you define an initial velocity at a single degree of freedom, the initial velocity at every other degree of freedom will be 0.0, potentially leading to conflicting initial conditions. In most cases, you should define initial conditions at every unconstrained degree of freedom in your model. If you define an initial condition for any degree of freedom at the pilot node of a rigid body (see Modeling Rigid Bodies in the *Contact Technology Guide* for the definition of rigid body), then the same initial condition must also be defined for the same degree of freedom on all other nodes of the rigid body.

After a solution has been performed, the specified initial conditions are overwritten by the actual solution and are no longer available. You must respecify them if you want to perform a subsequent analysis. You may want to keep a database file saved prior to the first solution for subsequent reuse.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you use the **CDWRITE (p. 293)** command to archive your model, initial displacements, temperatures, etc. specified via the **IC (p. 921)** command are not written to the archive file. Initial velocities and accelerations *are* written.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Define

Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Define

---

ICDELE

Deletes initial conditions at nodes.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Deletes all initial conditions previously specified with the **IC (p. 921)** or **ICROTATE (p. 925)** command at all nodes.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Initial Condit'n

Main Menu>Solution>Define Loads>Delete>Initial Condit'n

ICLIST, *NODE1*, *NODE2*, *NINC*, *Lab*

Lists the initial conditions.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2*, *NINC

List initial conditions for nodes *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and initial conditions for all selected nodes (**NSEL (p. 1266)**) are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Lab

Label identifying the initial condition to list out:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DISP

Displacements, temperature, etc. (default).

VELO

Velocities.

ACC

Accelerations.

Notes

Lists the initial conditions specified by the **IC (p. 921)** or **ICROTATE (p. 925)** command. Listing applies to all the selected nodes (**NSEL (p. 1266)**) and DOF labels. **ICLIST (p. 923)** is not the same as the **DLIST (p. 550)** command. All the initial conditions including the default conditions are listed for the selected nodes.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>List Picked

Main Menu>Solution>Define Loads>Apply>Initial Condit'n>List Picked

Utility Menu>List>Loads>Initial Conditions>On Picked Nodes

/ICLWID, *FACTOR*

Scales the line width of circuit builder icons.

GRAPHICS (p. 16): Scaling (p. 17)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FACTOR

Multiplication factor applied to the default line width (defaults to 1). The minimum is 1 and the maximum is 6.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ICROTATE, *NODE*, *OMEGA*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*, *Vx*, *Vy*, *Vz*, *ACCEL*

Specifies initial velocity at nodes as a sum of rotation about an axis and translation.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node at which the initial velocity is to be specified. If ALL, apply to all selected nodes (**NSEL (p. 1266)**). A component name may be input for *NODE*.

OMEGA

Scalar rotational velocity about the rotational axis.

X1, Y1, Z1

Coordinates (in the global Cartesian coordinate system) of the beginning point of the rotational axis vector.

X2, Y2, Z2

Coordinates (in the global Cartesian coordinate system) of the end point of the rotational axis vector.

Vx

Initial translational velocity in direction x of the nodal coordinate system.

Vy

Initial translational velocity in direction y of the nodal coordinate system.

Vz

Initial translational velocity in direction z of the nodal coordinate system.

ACCEL

Key to initialize acceleration due to centrifugal effects:

(blank)

Do not initialize acceleration (default).

CENT

Initialize acceleration due to centrifugal effects along with the initial velocity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **ICROTATE (p. 925)** command specifies initial velocity for all translational degrees of freedom of the specified nodes. The velocity value is a combination of velocity due to rotation about an axis and translation. The velocity at the node is calculated as:

$$v^N = v^{trans} + \omega \left[\frac{x^2 - x^1}{||x^2 - x^1||} \times (x^N - x^1) \right]$$

where

v^N = velocity of node *N* in the nodal coordinate system

v^{trans} = translational velocity input as [*VX*, *VY*, *VZ*]

ω = scalar angular velocity input as *OMEGA*

x^1 and x^2 denote the coordinates of points prescribing the beginning [*X1*, *Y1*, *Z1*] and end [*X2*, *Y2*, *Z2*] of the rotation axis

x^N denotes the coordinates of node *N*

All coordinates are input in the global Cartesian coordinate system, and the velocity due to rotation is then converted to the nodal coordinate system and added to the prescribed translation.

If *ACCEL* = CENT is specified, acceleration due to centrifugal effects is initialized as well. The acceleration at node a^N is initialized as:

$$a^N = \omega \left[\frac{x^2 - x^1}{||x^2 - x^1||} \right] \times \left(\omega \left[\frac{x^2 - x^1}{||x^2 - x^1||} \right] \times (x^N - x^1) \right)$$

The **ICROTATE (p. 925)** command is valid only for static analysis and full method transient analysis (**TIMINT (p. 2001)**,ON with **TRNOPT (p. 2017)**,FULL). The initial value is specified at the beginning of the first load step; that is, at time = 0.0.

The command calculates the nodal velocities and saves them in the database as if the **IC (p. 921)** command had been used to calculate these velocities. Thus, when the `Jobname.cdb` or `Jobname.db` file is written, the velocities prescribed by the **ICROTATE (p. 925)** command appear as **IC (p. 921)** commands. All assumptions, recommendations, and restrictions for the **IC (p. 921)** command are also true for the **ICROTATE (p. 925)** command.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/ICSCALE, *WN*, *FACTOR*

Scales the icon size for elements supported in the circuit builder.

GRAPHICS (p. 16): Scaling (p. 17)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

FACTOR

Factor applied to the default icon size (defaults to 1).

Notes

Scaling the icon size can provide better visualization of the circuit components when using the Circuit Builder (an interactive builder available in the Mechanical APDL GUI).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon

*IF, *VAL1*, *Oper1*, *VAL2*, *Base1*, *VAL3*, *Oper2*, *VAL4*, *Base2*

Conditionally causes commands to be read.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

VAL1

First numerical value (or parameter which evaluates to a numerical value) in the conditional comparison operation. *VAL1*, *VAL2*, *VAL3*, and *VAL4* can also be character strings (enclosed in quotes) or parameters for *Oper* = EQ and NE only.

Oper1

Operation label. A tolerance of 1.0E-10 is used for comparisons between real numbers:

EQ --

Equal (for *VAL1* = *VAL2*).

NE --

Not equal (for *VAL1* ≠ *VAL2*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LT --

Less than (for *VAL1* < *VAL2*).

GT --

Greater than (for *VAL1* > *VAL2*).

LE --

Less than or equal (for *VAL1* ≤ *VAL2*).

GE --

Greater than or equal (for *VAL1* ≥ *VAL2*).

ABLT --

Absolute values of *VAL1* and *VAL2* before < operation.

ABGT --

Absolute values of *VAL1* and *VAL2* before > operation.

VAL2

Second numerical value (or parameter which evaluates to a numerical value) in the conditional comparison operation.

Base1

Action based on the logical expression (*Oper1*) being true. If false, continue reading at the next line. This is conditional, except for the IF-THEN-ELSE constructs described below; any of the following constructs (through *Base1* = THEN) cause all subsequent fields to be ignored:

:*label* --

A user-defined label (beginning with a colon (:), 8 characters maximum). The command reader will skip (and wrap to the beginning of the file, if necessary) to the first line that begins with the matching :*label*.

Caution:

This label option may not be mixed with do-loop or if-then-else constructs.

STOP --

This action will cause an exit from the Mechanical APDL program at this line, unless running in interactive mode. In interactive mode, the program will not stop.

EXIT --

Exit the current do-loop (***EXIT (p. 707)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CYCLE --

Skip to the end of the current do-loop (***CYCLE (p. 462)**).

THEN --

Make this ***IF (p. 927)** an if-then-else construct (see below).

The following optional values determine the connection between the two logical clauses *Oper1* and *Oper2*

AND --

True if both clauses (*Oper1* and *Oper2*) are true.

OR --

True if either clause is true.

XOR --

True if either (but not both) clause is true.

VAL3

Third numerical value (or parameter which evaluates to a numerical value).

Oper2

Operation label. This will have the same labels as *Oper1*, except it uses *Val3* and *Val4*. A tolerance of 1.0E-10 is used for comparisons between real numbers.

VAL4

Fourth numerical value (or parameter value which evaluates to a numerical value).

Base2

Action based on the logical expression (*Oper1* and *Oper2*) being true. They will be the same values as *Base1*, except as noted.

Command Default

Read commands sequentially.

Notes

Conditionally causes commands to be read from a specific block or at a specific location. Twenty levels of nested ***IF (p. 927)** blocks are allowed. Jumping to a :*label* line is not allowed with keyboard entry. Jumping into, out of, or within a do-loop or an if-then-else construct to a :*label* line is not allowed. Using ***IF (p. 927)** interactively or from the command line prevents rereading the file to find a *label*. To do so, use batch mode or **/INPUT (p. 948)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 9: IF-THEN-ELSE Construct

***IF (p. 927)**,*VAL1*,*0per*,*VAL2*,THEN

***ELSEIF (p. 620)**,*VAL1*,*0per*,*VAL2*

***ELSEIF (p. 620)**,*VAL1*,*0per*,*VAL2*

***ELSE (p. 620)**

***ENDIF (p. 636)**

where "----" represents a block of any number of commands.

Any number of ***ELSEIF (p. 620)** clauses (or none) may be included (in the location shown). One ***ELSE (p. 620)** clause (at most) may be included (in the location shown).

The ***IF (p. 927)** command is executed by evaluating its logical expression. If it is true, the block of commands following it is executed. The construct is considered to be complete and the command following the ***ENDIF (p. 636)** is executed next.

If the logical expression is false, the next ***ELSEIF (p. 620)** command (if any) following the block is executed. The execution logic is the same as for ***IF (p. 927)**. The effect is that the logical expressions in the ***IF (p. 927)** and the ***ELSEIF (p. 620)** commands are sequentially tested until one is found to be true. Then the block of commands immediately following the expression is executed, which completes the execution of the if-then-else construct.

If all ***IF (p. 927)** and ***ELSEIF (p. 620)** expressions are false, the block following the ***ELSE (p. 620)** command is executed, if there is one. Only one block of commands (at most) is executed within the if-then-else construct.

If a batch input stream encounters an end-of-file during a false ***IF (p. 927)** condition, the Mechanical APDL run ends abnormally. You will need to terminate it externally (via the Linux "kill" function or the Windows task manager).

The ***IF (p. 927)**, ***ELSEIF (p. 620)**, ***ELSE (p. 620)**, and ***ENDIF (p. 636)** commands for each if-then-else construct must all be read from the same file (or keyboard).

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IGESIN, *Fname*, *Ext*, --

Transfers IGES data from a file into Mechanical APDL.

AUX15 (p. 63): IGES (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to CAD if *Fname* is blank.

--

Unused field.

Notes

Reads a file containing IGES data and transfers it into the Mechanical APDL database. The file transferred is the IGES Version 5.1, ASCII format file. IGES (Initial Graphics Exchange Specification) is a neutral format developed by the U.S. Dept. of Commerce, National Institute of Standards and Technology. No output transfer file is written because the transferred data is read directly into the Mechanical APDL database.

You can import multiple files into a single database, but you must use the same import option (set with the **IOPTN (p. 957)** command) for each file.

The **IOPTN (p. 957)** command sets the parameters for reading the file. Files read via the SMOOTH method (the only available method) use the standard database.

Menu Paths

Utility Menu>File>Import

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IGESOUT, *Fname*, *Ext*, --, *ATT*

Writes solid model data to a file in IGES Version 5.1 format.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to IGES if *Fname* is blank.

--

Unused field.

ATT

Attribute key:

0

Do not write assigned numbers and attributes of the solid model entities to the IGES file (default).

1

Write assigned numbers and attributes of solid model entities (keypoints, lines, areas, volumes) to the IGES file. Attributes include MAT, TYPE, REAL, and ESYS specifications as well as associated solid model loads and meshing (keypoint element size, number of line divisions and spacing ratio) specifications.

Notes

Causes the selected solid model data to be written to a coded file in the IGES Version 5.1 format. Previous data on this file, if any, are overwritten.

Keypoints that are not attached to any line are written to the output file as IGES entity 116 (Point).

Lines that are not attached to any area are written to the output file as either IGES Entity 100 (Circular Arc), 110 (Line), or 126 (Rational B-Spline Curve) depending upon whether the Mechanical APDL entity was defined as an arc, straight line, or spline.

Areas are written to the output file as IGES Entity 144 (Trimmed Parametric Surface).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volumes are written to the output file as IGES entity 186 (Manifold Solid B-Rep Object).

Solid model entities to be written must have all corresponding lower level entities selected (use **ALLSEL (p. 114)**,BELOW,ALL) before issuing command.

Concatenated lines and areas are not written to the IGES file; however, the entities that comprise the concatenated entities are written.

Caution:

Section properties assigned to areas, lines and other solid model entities are not maintained when the model is exported via **IGESOUT (p. 932)**.

Issuing **IGESOUT (p. 932)** after generating a beam mesh with orientation nodes causes the orientation keypoints specified for the line (**LATT (p. 1014)**) to no longer be associated with the line and are therefore not written out to the IGES file. The line does not recognize that orientation keypoints were ever assigned to it. Therefore, **IGESOUT (p. 932)** does not support (for beam meshing) any line operation relying on solid model associativity. (For example, meshing the areas adjacent to the meshed line, plotting the line that contains the orientation nodes, or clearing the mesh from the line that contains orientation nodes may not work as expected.) For more information about beam meshing, see Meshing Your Solid Model.

Menu Paths

Utility Menu>File>Export

/IMAGE, *Label*, *Fname*, *Ext*, --

Allows graphics data to be captured and saved.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Label specifying the operation to be performed:

CAPTURE

Capture the image from the graphics window to a new window.

RESTORE

Restore the image from a file to a new window.

SAVE

Save the contents of the graphic window to a file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELETE

Delete the window that contains the file.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

If no extension is specified, `bmp` will be used on Windows systems, and `img` will be used on Linux systems.

--

Unused field.

Menu Paths

This command cannot be accessed from a menu.

IMAGIN, *IR*, *IA*, - -, - -, *Name*, - -, - -, *FACTA*

Forms an imaginary variable from a complex variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

--,--

Unused fields.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- -, - -

Unused fields.

FACTA

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

This command forms a new variable from a complex variable by storing the imaginary part as the real part. The imaginary part can then be used in other operations. Used only with harmonic analyses (**ANTYPE (p. 162)**,HARMIC).

Complex variables are stored in two-column arrays with the real component stored in the first column and the imaginary component stored in the second column. This command extracts the value stored in the second column (that is, imaginary component). However, with harmonic analyses, all variables are stored in two-column arrays as complex variables. If the variable is not complex, then the same value is stored in both columns. This command will extract the variable in the second column of the array, even if this variable is not the imaginary component of a complex variable.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Imaginary Part

IMESH, *LAKY*, *NSLA*, *NTLA*, *KCN*, *DX*, *DY*, *DZ*, *TOL*

Generates nodes and interface elements along lines or areas.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LAKY

Copies mesh according to the following:

LINE or 1

Copies line mesh (default).

AREA or 2

Copies area mesh.

NSLA

Number that identifies the source line or area. This is the line or area whose mesh will provide the pattern for the interface elements. Mechanical APDL copies the pattern of the line or area elements through the area or volume to create the mesh of area or volume interface elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NTLA

Number that identifies the target line or area. This is the line or area that is opposite the source line or area specified by *NSLA*. Add *NTLA* to obtain the copied mesh from the source line or area.

KCN

Number that identifies the particular Mechanical APDL coordinate system.

DX, DY, DZ

Incremental translation of node coordinates in the active coordinate system (*DR*, *D* , *DZ* for cylindrical, and *DR*, *D* , *D* for spherical or toroidal). The source line or area coordinates + *DX*, *DY*, *DZ* = the target line or area coordinates. If left blank, Mechanical APDL automatically estimates the incremental translation.

TOL

Tolerance for verifying topology and geometry. By default, Mechanical APDL automatically calculates the tolerance based on associated geometries.

Notes

Generates nodes and interface elements along lines or areas. The **IMESH (p. 935)** command requires that the target line or area exactly match the source line or area. Also, both target and source lines or areas must be in the same area or volume. The area or volume containing the source line or area must be meshed before executing **IMESH (p. 935)**, while the area or volume containing the target line or area must be meshed after executing **IMESH (p. 935)**.

For three dimensional problems where *LAKY* = AREA, Mechanical APDL fills the interface layer as follows:

If source mesh consists of:	Mechanical APDL fills the interface layer with:
Quadrilateral elements	Hexahedral elements
Triangle elements	Degenerated wedge elements
Combination quadrilateral and triangle elements	Combination hexahedral and degenerated wedge elements

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Interface Mesh>2D Interface

Main Menu>Preprocessor>Meshing>Mesh>Interface Mesh>3D Interface

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IMMED, *KEY*

Allows immediate display of a model as it is generated.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Immediate mode key:

0

Display only upon request, that is, no immediate display (default with the GUI *off*).

1

Display immediately as model is generated (default with the GUI *on*).

Notes

The command enables you to control whether or not the model is displayed immediately as it is generated in an interactive session. Available only during an interactive session at a graphics display terminal. A valid graphics device name must first be specified (**/SHOW (p. 1785)**).

By default in the GUI, your model is immediately displayed in the Graphics window as you create new entities (such as areas, keypoints, nodes, elements, local coordinate systems, boundary conditions, etc.), referred to as *immediate mode* graphics.

Symbols (such as boundary conditions, local coordinate system triads, etc.) appear immediately and are present on subsequent displays unless you disable the appropriate symbol (via the GUI plot controls function or the appropriate graphics-specification command).

An immediate image is also scaled automatically to fit within the Graphics window. The new scaling is usually apparent on the automatic replot associated with immediate mode. To suppress automatic replot, issue **/UIS (p. 2045)**,REPLOT,0. (With automatic replot suppressed, the immediate image may not always be scaled correctly.)

An immediate display in progress should *not* be aborted with the usual system "break" feature (or else the Mechanical APDL session itself terminates). When you run Mechanical APDL interactively *without* using the GUI, immediate mode is off by default.

This command is valid only in PREP7.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Immediate Display

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INISTATE,*Action*,*Val1*,*Val2*,*Val3*,*Val4*,*Val5*,*Val6*,*Val7*,*Val8*,*Val9*

Defines initial-state data and parameters.

PREP7 (p. 22): Data Tables (p. 24)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Specifies action for defining or manipulating initial-state data:

SET

Use *Action* = SET to designate initial-state coordinate system, data type, and material type parameters. See "Command Specification for *Action* = SET" (p. 939).

DEFINE

Use *Action* = DEFINE to specify the actual state values, and the corresponding element, integration point, or layer information. See "Command Specifications for *Action* = DEFINE" (p. 940).

Use *Action* = DEFINE for function-based initial state. See "Command Specifications for *Action* = DEFINE (Function-Based Option)" (p. 942).

WRITE

Use *Action* = WRITE to write the initial-state values to a file when the **SOLVE (p. 1822)** command is issued. See "Command Specifications for *Action* = WRITE" (p. 943).

READ

Use *Action* = READ to read the initial-state values from a file. See "Command Specifications for *Action* = READ" (p. 944).

LIST

Use *Action* = LIST to read out the initial-state data. See "Command Specifications for *Action* = LIST" (p. 945).

DELETE

Use *Action* = DELE to delete initial-state data from a selected set of elements. See "Command Specifications for *Action* = DELETE" (p. 945)

Val1, Val2, ..., Val9

Input values based on the *Action* type.

Notes

INISTATE (p. 938) is available for current-technology elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The command can also be used with MESH200 (via the mesh-independent method for defining reinforcing) to apply an initial state to all generated reinforcing elements automatically. For more information, see Applying an Initial State to Reinforcing Elements in the *Structural Analysis Guide*.

Initial-state support for a given element is indicated in the documentation for the element under **Special Features**.

Initial-strain input (**INISTATE (p. 938)**,SET,DTYPE,EPEL) enables the nonlinear solver option automatically even if no nonlinear materials are involved.

The command does not support kinematic hardening material properties (such as **TB (p. 1899)**,PLASTIC,,,,BKIN) or the shape memory alloy material model (**TB (p. 1899)**,SMA).

INISTATE (p. 938) with elastic strain alone is not supported for gasket materials (**TB (p. 1899)**,GASK) and hyperelastic materials (**TB (p. 1899)**,HYPER, **TB (p. 1899)**,BB, **TB (p. 1899)**,AHYPER, **TB (p. 1899)**,CDM, **TB (p. 1899)**,EXPE).

INISTATE (p. 938) with initial stress alone is not supported for gasket materials (**TB (p. 1899)**,GASK).

INISTATE (p. 938) with plastic strain (which must include initial strain or stress, plastic strain, and accumulated plastic strain) does not support gasket materials (**TB (p. 1899)**,GASK), rate-dependent plasticity (**TB (p. 1899)**,RATE), and viscoelasticity (**TB (p. 1899)**,PRONY, **TB (p. 1899)**,SHIFT).

For more information about using the initial-state capability, see Initial State in the *Advanced Analysis Guide*.

Command Specification for *Action* = SET

INISTATE (p. 938), SET, *Val1*, *Val2*

Val1 **=**	***Val2*** **=**
CSYS	Coordinate system. *Val2* is an integer corresponding to the coordinate system: -2 = Element coordinate system -1 = Material coordinate system 0 = Global Cartesian coordinate system 0 - 10 = Any Mechanical APDL-defined coordinate system ≥11 = Any user-defined coordinate system ID
DTYP	Data type. *Val2* is the type of data that will be set on the subsequent **INISTATE (p. 938)**,DEFINE command: STRE = Stress data (default) EPEL = Strain data EPPL = Plastic strain data BSTR = Initial backstress PLEQ = Accumulated equivalent plastic strain

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val1 =	*Val2* =
	PLWK = Plastic strain energy density EPCR = Creep strain data PPRE = Pore pressure VOID = Void ratio SVAR = State variables RELD = Relative density uf*nn* = User-defined field *nn* (01 through 09)
MAT	Material type. *Val2* is the material ID. Specifying *Val2* = 0 disables material-based initial state and enables integration-point-based initial-state data.
NODE	Enable node-based initial state. When *Val2* = 1, all subsequent **INISTATE (p. 938)** commands use the node-based format. To disable node-based initial state, specify *Val2* = 0.
DATA	Input method. By default, the data is discrete at either the node- or element-integration point. Function-based initial state can be activated via the FUNC option.
STOE	*Val2* is an integer specifying the initial-stress-to-strain flag: 0 = Disable initial-stress to initial-strain conversion (default). 1 = Enable initial-stress to initial-strain conversion.

Notes

Action = SET specifies and modifies the environment into which you will define the initial-state data (via a subsequent **INISTATE (p. 938)**,DEFINE command). Otherwise, subsequent **INISTATE (p. 938)**,DEFINE data is input as initial stress data in the global Cartesian coordinate system.

Command Specifications for *Action* = DEFINE

INISTATE (p. 938),DEFINE,*ELID, EINT, KLAYER, PARMINT, Cxx, Cyy, Czz, Cxy, Cyz, Cxz*

ELID --

Element ID number when using element-based initial state. Defaults to current element selection.

Node number when using node-based initial state. Defaults to current node selection.

EINT --

Gauss integration point. Default = ALL or -1.

For node-based initial state (*Val2* = NODE), element ID number (if specified). The **INISTATE (p. 938)** command is applied only to the specified element (unlike the default behavior, where the command is applied to *all* selected elements containing the specified node).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Not valid for material-based initial-state data (*Val1* = MAT) or node-based initial state (*Val2* = NODE).

KLAYER **--**

Layer number (for layered solid/shell elements) or cell number for beam/pipe elements. Blank for other supported element types and material-based initial-state data. Default = ALL or -1.

ParmInt **--**

Section integration point within a layer, or cell-integration point for beams (typically four integration points). Default = ALL or -1. Not valid for material-based initial-state data (*Val1* = MAT) or node-based initial state (*Val2* = NODE).

Not valid for material-based initial-state data (*Val1* = MAT).

Not used for node-based initial state with elements that do not have a beam/pipe/shell section defined.

For node-based initial state with beams/pipes, values 1 through 4 can be used to specify the values at corner nodes within a cell.

For node-based initial state with layered sections, values can be specified at TOP, BOT, and MID, or left blank (ALL or -1).

Cxx,Cyy,Czz,Cxy,Cyz,Cxz **--**

Stress (S), strain (EPEL), or plastic strain (EPPL) values.

For initial backstress (BSTR), the bilinear kinematic hardening (**TB (p. 1899)**,PLASTIC,,,,BKIN) and Chaboche nonlinear kinematic hardening (**TB (p. 1899)**,CHABOCHE) material models are supported.

Bilinear kinematic hardening defines all six values (*Cxx*, *Cyy*, *Czz*, *Cxy*, *Cyz*, *Cxz*), similar to initial stress or initial plastic strain.

For the Chaboche model, you can define up to 12 values (*Action* = DEFINE), so you can use this method for two subchains of the model. The input format becomes:

Cxx1,Cyy1,Czz1,Cxy1,Cyz1,Cxz1,Cxx2,Cyy2,Czz2,Cxy2,Cyz2,Cxz2

If more than two subchains of the Chaboche model are required, the initial backstress definition is possible via *Action* = READ only. The `.ist` file supports up to five subchains of the Chaboche model. For five subchains of the Chaboche model, for example, the input format becomes:

Cxx1,Cyy1,Czz1,Cxy1,Cyz1,Cxz1,…,Cxx5,Cyy5,Czz5,Cxy5,Cyz5,Cxz5

For more information, see Applying Initial Backstress in the *Advanced Analysis Guide*. Also see Bilinear Kinematic Hardening and Nonlinear Kinematic Hardening in the Theory Reference

Notes

You can issue the **INISTATE (p. 938)** command repeatedly to define multiple sets of initial-state values. initial-state data can be specified according to elements, layers or integration points.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the initial-state parameters are being defined based on the material, (**INISTATE (p. 938)**,SET,MAT,*MATID*), *ELID* designates the element ID number and all subsequent values are ignored.

For coupled-field elements, the stresses to input must be Biot's effective stresses.

Command Specifications for *Action* = DEFINE (Function-Based Option)

INISTATE (p. 938),DEFINE,*ELID, EINT, --, --, FuncName, C1, C2, ..., C12*

ELID **--**

Element ID number when using element-based initial state. Defaults to current element selection.

Node number when using node-based initial state. Defaults to current node selection.

EINT **--**

Gauss integration point (defaults to ALL). Not valid for material-based initial-state data (*Val1* = MAT) or node-based initial state (*Val2* = NODE).

(Blank) --

Reserved for future use.

(Blank) --

Reserved for future use.

FuncName **--**

LINX | LINY | LINZ. Apply initial-state data as a linear function of location based on the X axis (LINX), Y axis (LINY), or Z axis (LINZ) in the coordinate system specified via the **INISTATE (p. 938)**,SET,CSYS command. Default coordinate system: CSYS,0 (global Cartesian).

C1, C2, ..., C12 **--**

For *FuncName* with tensors, each component uses two values. SXX = *C1* + X**C2*, SYY = *C3* + X**C4*, and so on. Specify 12 values (for the six tensor components).

For *FuncName* with scalars, only two values *C1* and *C2* (*VALUE* = *C1* + X**C2*) are necessary to apply the initial state.

Notes

You can issue **INISTATE (p. 938)** repeatedly with the function-based option to define multiple sets of initial-state values. Initial-state data can be specified according to elements or integration points.

For coupled-field elements, the stresses to input must be Biot's effective stresses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Specifications for *Action* = WRITE

INISTATE,WRITE,*FLAG, , , , CSID, Dtype*

FLAG **--**

Set this value to 1 to generate the initial-state file, or 0 to disable initial-state file generation.

CSID **--**

Determines the coordinate system for the initial state:

0 (default)

Write in global Cartesian coordinate system for solid elements.

-1 (or MAT)

Write in material coordinate system

-2 (or ELEM)

Write in element coordinate system for link, beam, and layered elements.

Dtype **--**

Sets the data type to be written in the `.ist` file:

S

Output stresses.

EPEL

Output elastic strain.

EPPL

Output plastic strain.

PLEQ

Output equivalent plastic strain.

PLWK

Output plastic strain energy density.

EPCR

Output creep strain.

PPRE

Initial pore pressure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VOID

Initial void ratio.

SVAR

State variables.

BSTR

Initial backstress.

Notes

Default is 0 for solid elements and -2 for link, beam, and shell elements.

State variables are always written to the `.ist` file in the material coordinate system.

Only the three in-plane stresses for the top and bottom surfaces are written.

For coupled-field elements, the stresses written out are Biot's effective stress values.

Initial pore pressure and void ratio are available for the coupled pore-pressure elements (CPT*nnn*) only: CPT212, CPT213, CPT215, CPT216, and CPT217.

Command Specifications for *Action* = READ

INISTATE (p. 938), READ, *Fname, Ext, Path, MeshIndMethod*

Reads initial-state data from a standalone initial-state (.ist) file of the specified name (*Fname*) and filename extension (*Ext*), located in the specified path (*Path*). The initial-state file must be in a comma-delimited ASCII file format, consisting of individual rows for each stress/strain item, with each row consisting of columns separated by commas.

Use *Action* = READ to apply complex sets of initial-state data to various elements, cells, layers, sections, and integration points. This option is available for element-integration-point-based initial-state data and node-based initial-state data.

Mapping to nodes may offer better performance when many substeps are involved; however, only location support is available. Mapping to element-integration points supports additional field variables TIME, FREQ and TEMP and generally uses less memory.

For other non-user-defined field variables (such as initial stress or strain), initial state is evaluated only at the first substep in the first load step.

MeshIndMethod **--**

Mesh-Independent method `.ist` read options:

0 or DEFA -- Standard (mesh-dependent) initial state `.ist` file (default).

MAPN -- Map to nodes internally when applying the initial state.

MAPI -- Map to element-integration points.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DOBJ -- Do not use `.ist` data in the finite element solution. (Use this option if converting initial-stress data to a traction load (p. 342).)

Command Specifications for *Action* = LIST

INISTATE, LIST, *ELID*

If using the standard method for applying initial-state data,specify *ELID* = element ID number to list initial-state data for elements. If *ELID* is unspecified, *all* initial-state data for all selected elements are listed.

If using the mesh-independent method, specify *ELID* = MIND to list initial-state data.

Command Specifications for *Action* = DELETE

INISTATE, DELE, *ELID*

If using the standard method, specify *ELID* = element ID number to delete initial-state data for elements. If *ELID* is unspecified, *all* initial-state data for all selected elements are deleted.

If using the mesh-independent method, specify *ELID* = MIND to delete initial-state data.

Menu Paths

This command cannot be accessed from a menu.

---

*INIT, *Name*, *Method*, *Val1*, *Val2*, *Val3*

Initializes a vector or matrix.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name

Vector or matrix which will be initialized. This can be a vector (created by the ***VEC (p. 2085)** command), a dense matrix (created by the ***DMAT (p. 551)** command), or a sparse matrix (created by the ***SMAT (p. 1801)** command).

Method

Initialization method to use:

ZERO --

Fill the vector/matrix with zeros (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONST --

Fill the vector/matrix with a constant value.

RAND --

Fill the vector/matrix with random values.

DIAG --

Fill the *n*th diagonal of the matrix with a constant value. Other values are not overwritten. For this option, *Name* must be a dense matrix.

ADIAG --

Fill the *n*th anti-diagonal of the matrix with a constant value. Other values are not overwritten. For this option, *Name* must be a dense matrix.

CONJ --

Take the complex conjugate of the values in the vector/matrix (no change for non-complex values).

FILTER --

Initialize a subset of values of a vector using a filtering vector. For this option, *Name* must be a vector.

Val1*, *Val2*, *Val3

Additional input. The meaning of *Val1* through *Val3* will vary depending on the specified *Method*. See details below.

The following *Valx* fields are used with *Method* = CONST:

Val1

The real part of the constant value to use (default = 0).

Val2

The imaginary part of the constant value to use (default = 0). Required only for a complex vector/matrix.

The following *Valx* fields are used with *Method* = DIAG or *Method* = ADIAG:

Val1

The number of the diagonal to fill. A zero value (which is the default) indicates the main diagonal (or main anti-diagonal). A positive value indicates an upper diagonal; a negative value indicates a lower diagonal.

Val2

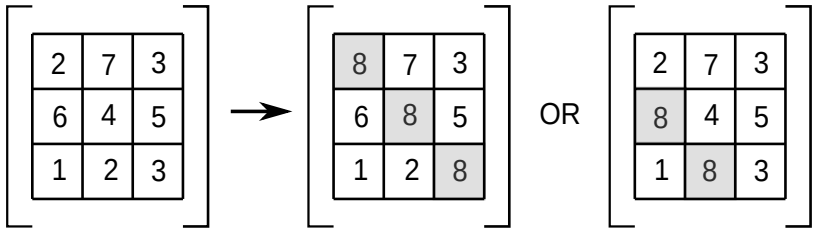
The real part of the constant value to use (default = 1).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val3

The imaginary part of the constant value to use (default = 0). Required only for a complex matrix.

The following example demonstrates *Method* = DIAG:



Original Matrix, M1 *INIT,M1,DIAG,0,8 *INIT,M1,DIAG,-1,8

The following *Valx* fields are used with *Method* = FILTER:

Val1

The name of an existing integer vector (created by ***VEC (p. 2085)**) to be used as the filter vector. The values in this vector indicate the locations in the *Name* vector that are to be initialized to *Val2* (real value) and *Val3* (imaginary value, if applicable). Location values higher than the dimension of the original vector are ignored.

Val2

The real part of the value used for initialization (default = 0).

Val3

The imaginary part of the value used for initialization (default = 0); applicable only if the *Name* vector contains complex values.

Notes

This command initializes a previously defined vector (***VEC (p. 2085)**), dense matrix (***DMAT (p. 551)**), or sparse matrix (***SMAT (p. 1801)**).

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/INPUT,*Fname*,*Ext*,*Dir*,*LINE*,*LOG*

Switches the input file for the commands that follow.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to the current Jobname if *Ext* is specified.

Ext

Filename extension (eight-character maximum).

Dir

Directory path (64 characters maximum). Defaults to current directory.

LINE

A value indicating either a line number in the file or a user-defined label in the file from which to begin reading the input file.

(blank), 0, or 1

Begins reading from the top of the file (default).

LINE_NUMBER

Begins reading from the specified line number in the file.

:*label*

Begins reading from the first line beginning with the matching user-defined label :*label* (beginning with a colon (:), 8 characters maximum).

LOG

Indicates whether secondary input from this command should be recorded in the command log (File.log) and the database log:

0

Record only the **/INPUT (p. 948)** command on the log (default).

1

Record commands in the specified secondary file as they are executed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Switches the input file for the next commands. Commands are read from this file until an end-of-file or another file switching directive is read. An end-of-file occurs after the last record of the file or when a **/EOF (p. 645)** command is read. An automatic switch back one level (to the previous file) occurs when an end-of-file is encountered. Twenty levels of nested file switching are allowed. Note that files including ***DO (p. 567)**, ***USE (p. 2059)**, ***ULIB (p. 2051)**, and the "Unknown Command" Macro have less nesting available because each of these operations also uses a level of file switching. For an interactive run, a **/INPUT (p. 948)**,TERM switches to the terminal for the next input. A **/EOF (p. 645)** read from the terminal then switches back to the previous file. A **/INPUT (p. 948)** (with a blank second field) switches back to the primary input file.

Setting *LOG* = 1 on **/INPUT (p. 948)** causes all commands read from the specified file to be recorded in the command log (`File.log`) and the internal database command log (**LGWRITE (p. 1048)**). This option is recommended if the log file will be used later. The *LOG* = 1 option is only valid when the **/INPUT (p. 948)** occurs in the primary input file. Using *LOG* = 1 on a nested **/INPUT (p. 948)** or on a **/INPUT (p. 948)** within a do-loop will have no effect (that is, commands in the secondary input file are not written to the command log).

The *Dir* option is optional as the directory path can be included directly in *Fname*.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Read Input from

---

*INQUIRE,*Obj*,*Property*,*Var1*

Retrieves properties of an existing APDL Math object.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Obj

Name of the vector or matrix of interest.

Property

Object property to get:

DIM1

First dimension of a matrix, or size of a vector.

DIM2

Second dimension of a matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Var1

Name of the resulting parameter that contains the property value.

Notes

The following example demonstrates using ***INQUIRE (p. 949)** to get the number of rows and columns of an existing matrix.

```
*SMAT,K,D,IMPORT,FULL,file.full,STIFF  ! Import the stiffness matrix from an existing FULL file
*INQUIRE,K,DIM1,NROW                  ! Get the first dimension of the stiffness matrix
*INQUIRE,K,DIM2,NCOL                  ! Get the second dimension of the stiffness matrix
/COM, K matrix size: %NROW% x %NCOL%
```

Menu Paths

This command cannot be accessed from a menu.

/INQUIRE,*StrArray*,*FUNC*

Returns system information to a parameter.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

StrArray

Name of the "string array" parameter that will hold the returned values. String array parameters are similar to character arrays, but each array element can be as long as 248 characters. If the string parameter does not exist, it will be created.

FUNC

Specifies the type of system information returned:

LOGIN --

Returns the pathname of the login directory on Linux systems or the pathname of the default directory (including drive letter) on Windows systems.

DOCU --

Returns the pathname of the Mechanical APDL docu directory.

APDL --

Returns the pathname of the Mechanical APDL APDL directory.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PROG --

Returns the pathname of the Mechanical APDL executable directory.

AUTH --

Returns the pathname of the directory in which the license file resides.

USER --

Returns the name of the user currently logged-in.

DIRECTORY --

Returns the pathname of the current directory.

JOBNAME --

Returns the current *Jobname*.

RSTDIR --

Returns rst directory (**FILE (p. 752)** command).

RSTFILE --

Returns rst file name (**FILE (p. 752)** command).

RSTEXT --

Returns rst file extension (**FILE (p. 752)** command).

PSEARCH --

Returns path used for "unknown command" macro (**/PSEARCH (p. 1535)** command).

OUTPUT --

Returns the current output file name (**/OUTPUT (p. 1334)** command).

Returning the Value of an Environment Variable to a Parameter

If *FUNC*=ENV, the command format is **/INQUIRE (p. 950)**,*StrArray*,ENV,*ENVNAME*,*Substring*. In this instance, ENV specifies that the command should return the value of an environment variable. The following defines the remaining fields:

ENVNAME

Specifies the name of the environment variable.

Substring

If *Substring* = 1, the first substring (up to the first colon (:)) is returned. If *Substring* = 2, the second substring is returned, etc. For Windows platforms, the separating character is semicolon (;). If this argument is either blank or 0, the entire value of the environment variable is returned.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Returning the Value of a Title to a Parameter

If *FUNC* = TITLE, the command format is **/INQUIRE (p. 950)**,*StrArray*,TITLE,*Title_num*. In this context, the value of *Title_num* can be blank or 1 through 5. If the value is 1 or blank, the title is returned. If the value is 2 through 5, a corresponding subtitle is returned (2 denoting the first subtitle, and so on).

Returning Information About a File to a Parameter

The **/INQUIRE (p. 950)** command can also return information about specified files within the file system. For these capabilities, the format is **/INQUIRE (p. 950)**,*Parameter*,*FUNC*,*Fname*, *Ext*, --. The following defines the fields:

Parameter

Name of the parameter that will hold the returned values.

FUNC

Specifies the type of file information returned:

EXIST --

Returns a 1 if the specified file exists, and 0 if it does not.

DATE --

Returns the date stamp of the specified file in the format *yyyymmdd.hhmmss*.

SIZE --

Returns the size of the specified file in MB.

WRITE --

Returns the status of the write attribute. A 0 denotes no write permission while a 1 denotes write permission.

READ --

Returns the status of the read attribute. A 0 denotes no read permission while a 1 denotes read permission.

EXEC --

Returns the status of the execute attribute (this has meaning only on Linux). A 0 denotes no execute permission while a 1 denotes execute permission.

LINES --

Returns the number of lines in an ASCII file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Notes

The **/INQUIRE (p. 950)** command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

INRES,*Item1*,*Item2*,*Item3*,*Item4*,*Item5*,*Item6*,*Item7*,*Item8*

Identifies the data to be retrieved from the results file.

POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item1,Item2,Item3,...,Item8

Data to be read into the database from the results file. May consist of any of the following labels:

ALL

All solution items (default).

BASIC

NSOL, RSOL, NLOAD, STRS, FGRAD, and FFLUX items.

NSOL

Nodal DOF solution.

RSOL

Nodal reaction loads.

ESOL

Element solution items (includes all of the following):

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NLOAD

Element nodal loads.

STRS

Element nodal stresses.

EPEL

Element elastic strains.

EPTH

Element thermal, initial, and swelling strains.

EPPL

Element plastic strains.

EPCR

Element creep strains.

FGRAD

Element nodal gradients.

FFLUX

Element nodal fluxes.

MISC

Element miscellaneous data (SMISC and NMISC).

NAR

Nodal-averaged result items (includes all of the following):

NDST

Nodal-averaged stresses.

NDEL

Nodal-averaged elastic strains.

NDPL

Nodal-averaged plastic strains.

NDCR

Nodal-averaged creep strains.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NDTH

Nodal-averaged thermal and swelling strains.

Notes

Identifies the type of data to be retrieved from the results file for placement into the database through commands such as **SET (p. 1724)**, **SUBSET (p. 1871)**, and **APPEND (p. 173)**. **INRES (p. 953)** is a companion command to the **OUTRES (p. 1336)** command controlling data written to the database and the results file. Since the **INRES (p. 953)** command can only flag data that has already been written to the results file, care should be taken when using the **OUTRES (p. 1336)** command to include all data you wish to retrieve for postprocessing later on.

The *Item* = ALL option includes all NAR items. For more information, see Nodal-Averaged Results in the *Element Reference*.

Menu Paths

Main Menu>General Postproc>Data & File Opts

INRTIA

Specifies "Inertial loads" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Loads>Inertia Loads

Utility Menu>List>Status>Solution>Inertia Loads

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INT1, *IR*, *IY*, *IX*, - - , *Name*, - - , - - , *FACTA*, *FACTB*, *CONST*

Integrates a variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result. Table values represent integrated sum of *IY* to current table position of *IX*.

IY, IX

Integrate variable *IY* with respect to *IX*.

--

Unused field.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --

Unused fields.

FACTA, FACTB

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

CONST

Initial value.

Notes

Integrates variables according to the operation:

$$IR = \int (FACTA \times IY)\, d(FACTB \times IX) + CONST$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Integrate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INVOPT, *Option*

Enables or disables inverse solving for the current load step.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Enables or disables inverse solving for a load step:

ON

Enable.

OFF

Disable and revert to forward solving (default).

Notes

Option = ON is valid only at the first load step of a static analysis. Large-deflection effects must be enabled (**NLGEOM (p. 1231)**,ON). The unsymmetric solver (**NROPT (p. 1261)**,UNSYM) is required and the program selects it automatically.

After issuing **INVOPT (p. 957)**,ON, inverse solving remains in effect until **INVOPT (p. 957)**,OFF is issued. The solution then reverts to traditional forward solving (default).

This command cannot be issued during a restart. *Option* can only be changed between load steps.

For more information, see Nonlinear Static Analysis with Inverse Solving in the *Structural Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

IOPTN, *Lab*, *VAL1*

Controls options relating to importing a model.

AUX15 (p. 63): IGES (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Label identifying the import option. The meaning of *VAL1* varies depending on *Lab*.

STAT

List overall status of import facilities, including current option values. *VAL1* is ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEFA

Set default values for all import options. *VAL1*is ignored.

MERG

Entity merge option. *VAL1* can be:

YES

Automatic merging is performed (default).

NO

No merging of entities.

SOLID

Solid option. *VAL1* can be:

YES

Solid is created automatically (default).

NO

No solid created.

GTOLER

Entity merging tolerance. If IGES = SMOOTH, the GTOLER,*VAL1* can be:

DEFA

Use system defaults (default).

FILE

Use tolerance from the imported file.

n

A user-specified tolerance value.

IGES

IGES import option. *VAL1* can be:

STAT

List status of IGES related options in the output window.

SMOOTH (or RV52)

Use more robust IGES revision 5.2 import function (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SMALL

Small areas option. *VAL1* can be:

YES

Small areas are deleted (default).

NO

Small areas are retained.

VAL1

Additional input value as described under each *Lab* option.

Command Default

Merging is performed during the IGES transfer with no global solid model tolerance (GTOLER) used.

Notes

Controls various options during a model file transfer. A global solid model tolerance (GTOLER) can be specified.

The SMALL,YES option (default) delete small areas and can cause geometrical inconsistencies that could cause the import process to abort. Retaining the small areas increases processor time and memory usage.

The data is stored in the standard Mechanical APDL graphics database.

The IGES,SMOOTH (default) option is capable of reading in any rational B-spline curve entity (type 126), or rational B-spline surface entity (type 128) with a degree less than or equal to 20. Attempts to read in B-spline curve or surface entities of degree higher than 20 may result in error messages.

If you issue the **/CLEAR (p. 350)** command, the **IOPTN (p. 957)** settings return to their defaults.

For MERG,YES, merging of coincident geometry items is performed automatically when the **IGESIN (p. 931)** command is issued (that is, an internal **NUMMRG (p. 1295)**,KP command is issued). The model is merged with the *consideration tolerance* (*TOLER* on **NUMMRG (p. 1295)**) set equal to 0.75 * the shortest distance between the endpoints of any active line. See the **NUMMRG (p. 1295)** command for more information about the tolerances. In most cases, the default merging is appropriate. Use the **IOPTN (p. 957)** command when you want to:

- Disable merging operations.
- Override the default merging and specify a global solid model tolerance value (GTOLER).
- Disable the automatic creation of solids (SOLID).

The **IOPTN (p. 957)** command should be issued *before* the **IGESIN (p. 931)** command. You cannot change these options after your model has been imported or created. If you must change the options:

1. Clear the database (**/CLEAR (p. 350)**) or exit and restart the program.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the correct options.

3. Reimport or recreate the model.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Import

IRLF,*KEY*,*PrintFreq*,*RampKey*

Specifies that inertia relief calculations are to be performed.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Calculation key:

0

No inertia relief calculations.

1

Counterbalance loads with inertia relief forces.

-1

Precalculate masses for summary printout only (no inertia relief).

PrintFreq

Frequency at which inertia relief information is printed during substeps in a nonlinear static or eigenvalue buckling analysis. This value must be a positive integer, 0, or ALL, as described below.

0

No inertia relief information is printed during the Newton Raphson substeps (default).

n

Prints inertia relief information every *n*th substep of each load step.

ALL

Prints inertia relief information every substep.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RampKey

Key to control ramping of rigid body accelerations. See notes for steps to achieve convergence when turning on the inertia relief option in the second or later load step of a nonlinear analysis (p. 962).

0

(Default) Rigid body acceleration is stepped and ramped together with total external loads following specifications set using the **KBC (p. 970)** command. Normally, the inertia relief option is turned on at the first load step.

1

Ramp the rigid body accelerations alone following specifications set by the **KBC (p. 970)** command. This option is only used for better convergence when the inertia relief load step is nonlinear and the inertia relief option is turned on from a later load step (other than the first). It works with **KBC (p. 970)**,0 specified and no new loads added to the current load step. After the current load step, *RampKey* is automatically set to 0. See Turning On the Inertia Relief Option in the Second or Later Load Step (p. 962).

Command Default

No inertia relief calculations.

Notes

The **IRLF (p. 960)** command specifies that the program is to calculate accelerations to counterbalance the applied loads (inertia relief). Displacement constraints on the structure should be only those necessary to prevent rigid-body motions (3 are needed for a 2D structure and 6 for a 3D structure). If the minimum number of displacement constraints are applied and it is a linear static analysis, the sum of the reaction forces at the constrained points will be zero. However, if it is a nonlinear static analysis (**NLGEOM (p. 1231)**,ON), the sum of reaction forces at the constrained points will be approximately zero due to the error introduced by nonlinear iterations. Accelerations are calculated from the element mass matrices and the applied forces. Data needed to calculate the mass (such as density) must be input. Both translational and rotational accelerations may be calculated.

This option applies to the following analyses: static (linear or nonlinear), linear perturbation static, and buckling (see Including Inertia Relief Calculations in the *Basic Analysis Guide* and Supported Analysis Types for Inertia Relief in the *Theory Reference*). Elements that operate in the nodal coordinate system and axisymmetric or generalized plane strain elements are not allowed. Symmetry models are not valid for inertia relief analysis. Models with both 2D and 3D element types are not recommended.

Loads may be input as usual. Displacements and stresses are calculated as usual.

An automatic inertia relief capability is available for linear static analysis only. See the **AIRL (p. 111)** command for details.

Use **IRLIST (p. 962)** to print inertia relief calculation results. Note that **IRLIST (p. 962)** can only be issued at the end of the analysis, right after the **SOLVE (p. 1822)** command in the nonlinear case. To print information during substeps in a nonlinear static or eigenvalue buckling analysis, set *PrintFreq* as described above. The mass and moment of inertia summary printed before the solution is accurate (because of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the additional pre-calculations required for inertia relief). See Inertia Relief in the *Mechanical APDL Theory Reference* for calculation details. See also the *Structural Analysis Guide* for procedural details.

Turning On the Inertia Relief Option in the Second or Later Load Step

If the inertia relief option is turned on in the second or later load step in a nonlinear static analysis, follow these precautionary steps for easy nonlinear convergence:

1. At the second or higher load step, where **IRLF (p. 960)**,1 is issued the first time, do not apply new external loads.
2. Set *Rampkey* = 1 at this load step and **KBC (p. 970)**,0 so that the rigid body accelerations are ramped.
3. Use the **NSUBST (p. 1287)** or **DELTIM (p. 515)** command to allow more substeps in this load step.

When a superelement (MATRIX50) is present in the model, any DOF constraints that you need to apply (**D (p. 483)**) on a degree of freedom (DOF) belonging to the superelement must be applied in the use pass of the MATRIX50 element (*not* in the generation pass). The command has no effect in the generation pass of a substructure. In the expansion pass, precalculation of masses for summary printout (**IRLF (p. 960)**,-1) occurs only on elements that are part of the substructure.

This command is also valid in PREP7.

Example Usage

Example command inputs for including inertia relief calculations in a static analysis with geometric nonlinearity, a linear perturbation static analysis, and a linear perturbation buckling analysis are found in Including Inertia Relief Calculations in the *Basic Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Inertia Relief

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Incl Mass Summry

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Inertia Relief

Main Menu>Solution>Load Step Opts>Output Ctrls>Incl Mass Summry

---

IRLIST

Prints inertia relief summary table.

POST1 (p. 51): Listing (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Prints the inertia relief summary data, including the mass summary table, the total load summary table, and the inertia relief summary table resulting from the inertia relief calculations. These calculations are performed in the solution phase (**SOLVE (p. 1822)**) as specified by the **IRLF (p. 960)** or **AIRL (p. 111)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inertia relief output is stored in the database rather than in the results file (`Jobname.rst`). When you issue **IRLIST (p. 962)** or **AIRL (p. 111)**, Mechanical APDL pulls the information from the database, which contains the inertia relief output from the most recent solution (**SOLVE (p. 1822)**).

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

---

*ITENGINE,*Type*,*EngineName*,*PrecondName*,*Matrix*,*RhsVector*,*SolVector*,*MaxIter*,*Toler*

Performs a solution using an iterative solver.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Type

Specifies the algorithm to be used:

PCG --

Preconditioned conjugate gradient (default).

EngineName

Name used to identify this iterative solver engine. Must be specified.

PrecondName

Linear solver engine name (***LSENGINE (p. 1077)**) identifying the factored matrix to be used as the preconditioner.

Matrix

Name of the matrix to solve.

RhsVector

Matrix (load vector) name.

SolVector

Solution vector name. If non-zero, it will be taken as the initial vector for the iterative process.

MaxIter

Maximum number of iterations allowed. Default is 2 times the number of rows in the matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Toler

Convergence tolerance. Default is 1.0E-8.

Notes

This command solves Ax = b using a preconditioned conjugate gradient algorithm. It uses an existing factored system as the preconditioner. This solution method is useful if an existing matrix has been solved and minor changes have been made to the matrix.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

J Commands

JPEG, *Keyword*, *OPT*

Provides JPEG file export for Mechanical APDL displays.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Keyword

Specifies various JPEG file export options.

QUAL

If *Keyword* = QUAL, then *OPT* is an integer value defining the JPEG quality index on an arbitrary scale ranging from 1 to 100. The default value is 75.

ORIENT

If *Keyword* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

COLOR

If *Keyword* = COLOR, then *OPT* will determine the color depth of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

TMOD

If *Keyword* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

DEFAULT

If *Keyword* = DEFAULT, then all of the default values, for all of the Keyword parameters listed above, are active.

OPT

OPT can have the following names or values, depending on the value for *Keyword* (see above).

1 to 100

If *Keyword* = QUAL, a value between 1 and 100 will determine the quality index of the JPEG file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Horizontal, Vertical

If *Keyword* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

0,1,2

If *Keyword* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

1,0

If *Keyword* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

Menu Paths

Utility Menu>PlotCtrls>HardCopy>ToFile

JSOL, *NVAR*, *ELEM*, *ITEM*, *COMP*, *Name*

Specifies result items to be stored for the joint element.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number or name assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR (p. 1302)**) while the name can be an eight-byte character string. Overwrites any existing results for this variable.

ELEM

Element number for which to store results.

Item

Label identifying the item. Valid item labels are shown in Table 191: JSOL - Valid Item and Component Labels (p. 967) below.

Comp

Component of the *Item* (if required). Valid component labels are shown in Table 191: JSOL - Valid Item and Component Labels (p. 967) below.

Name

Thirty-two character name identifying the item on printouts and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is valid for the MPC184 joint elements. The values stored are for the free or unconstrained degrees of freedom of a joint element. Relative reaction forces and moments are available only if stiffness, damping, or friction is associated with the joint element.

Table 191: JSOL - Valid Item and Component Labels

Item	Comp	Description
U	X, Y, Z	x, y, or z relative displacement.
ROT	X, Y, Z	x, y, or z relative rotation.
VEL	X, Y, Z	x, y, or z relative linear velocity.
OMG	X, Y, Z	x, y, or z relative angular velocity.
ACC	X, Y, Z	x, y, or z relative linear acceleration.
DMG	X, Y, Z	x, y, or z relative angular acceleration.
RF	X, Y, Z	Relative reaction forces in the local x, y, or z direction.
RM	X, Y, Z	Relative reaction moments in the local x, y, or z direction.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

K Commands

K, *NPT*, *X*, *Y*, *Z*

Defines a keypoint.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NPT

Reference number for keypoint. If zero, the lowest available number is assigned (**NUMSTR (p. 1301)**).

X, Y, Z

Keypoint location in the active coordinate system (may be R, θ, Z or R, θ, Φ). If *X* = P, graphical picking is enabled and all other fields (including *NPT*) are ignored (valid only in the GUI).

Notes

Defines a keypoint in the active coordinate system (**CSYS (p. 441)**) for line, area, and volume descriptions. A previously defined keypoint of the same number will be redefined. Keypoints may be redefined only if it is not yet attached to a line or is not yet meshed. Solid modeling in a toroidal system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>In Active CS

Main Menu>Preprocessor>Modeling>Create>Keypoints>On Working Plane

KATT, *MAT*, *REAL*, *TYPE*, *ESYS*

Associates attributes with the selected, unmeshed keypoints.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT, REAL, TYPE, ESYS

Material number, real constant set number, type number, and coordinate system number to be associated with selected, unmeshed keypoints.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Keypoints subsequently generated from the keypoints will also have these attributes. These element attributes will be used when the keypoints are meshed. If a keypoint does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT (p. 1111)**, **REAL (p. 1579)**,**TYPE (p. 2036)**, and **ESYS (p. 685)** command settings. Reissue the **KATT (p. 969)** command (before keypoints are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association.

If any of the arguments *MAT*, *REAL*, *TYPE*, or *ESYS* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, Mechanical APDL can proceed with a keypoint meshing operation even when no logical element type has been assigned via **KATT (p. 969)**,,,*TYPE* or **TYPE (p. 2036)**. For more information, see the discussion on setting element attributes in Meshing Your Solid Model in the *Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Keypoints

Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked KPs

KBC, *KEY*, *OMGSQRDKEY*

Specifies ramped or stepped loading within a load step.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Ramping key:

0

Loads are linearly interpolated (ramped) for each substep from the values of the previous load step to the values of this load step. This is the default value.

1

Loads are step changed (stepped) at the first substep of this load step to the values of this load step (that is, the same values are used for all substeps). Useful for rate-dependent behavior (for example, creep, viscoplasticity, etc.) or transient load steps only.

OMGSQRDKEY

Key for the interpolation of the rotational velocity loading (only supported when *KEY* = 0):

0

Rotational velocities are linearly interpolated. This is the default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

A quadratic interpolation is used for the rotational velocities (**OMEGA (p. 1324)**, **CMOMEGA (p. 368)**, and **CMROTATE (p. 373)**). All other loads are interpolated linearly.

Command Default

The loading is ramped for:

- static analysis (**ANTYPE (p. 162)**,STATIC)
- transient analysis with transient effects excluded (**ANTYPE (p. 162)**,TRANS and **TIMINT (p. 2001)**,OFF)
- harmonic analysis (**ANTYPE (p. 162)**,HARM)

The loading is step-applied for transient analysis with transient effects included (**ANTYPE (p. 162)**,TRANS and **TIMINT (p. 2001)**,ON; **TIMINT (p. 2001)**,ON is the default for transient analysis).

Notes

Specifies whether loads applied to intermediate substeps within the load step are to be stepped or ramped. Used only if *DTIME* on the **DELTIM (p. 515)** command is less than the time span or, conversely, if *NSBSTP* on the **NSUBST (p. 1287)** command is greater than one. Flags (FSI, MXWF, MVDI, etc.) are always stepped.

Changing the ramping *KEY* (that is, switching between ramped and stepped boundary conditions) between load steps is not recommended.

For ramped loading (**KBC (p. 970)**,0), when a load is applied for the first time, it is interpolated from zero to the value of the current load step, and not from the initial condition or value of the degree of freedom from the previous load step.

Spatially varying tabular loads or boundary conditions do not support direct ramping or stepping options and, instead, apply their full values according to the supplied tabular functions regardless of the **KBC (p. 970)** setting.

Regardless of the **KBC (p. 970)** setting, any tabular load is applied as step change. This is the case, for example, for a static or harmonic cyclic symmetry analysis with a load that varies by sector (**CYCOPT (p. 466)**, LDSECT). Note that when tabular and non-tabular loads are present in the same analysis, the non-tabular loads are ramped or stepped according to the **KBC (p. 970)** setting.

Irrespective of the **KBC (p. 970)** setting, loads are usually step-removed. See Stepping or Ramping Loads in the *Basic Analysis Guide* for more information.

It is sometimes difficult to obtain successful convergence with stepped loading in a nonlinear transient problem. If divergence is encountered, determine if stepped loading was used by default, then determine if it is appropriate for the analysis.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substeps
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>HOWP and Substeps
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substeps
Main Menu>Solution>Analysis Type>Sol'n Controls>Transient
Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substeps
Main Menu>Solution>Load Step Opts>Time/Frequenc>HOWP and Substeps
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step
Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substeps

---

KBETW, *KP1*, *KP2*, *KPNEW*, *Type*, *VALUE*

Creates a keypoint between two existing keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KP1

First keypoint. If *KP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KP2

Second keypoint.

KPNEW

Number assigned to the new keypoint. Defaults to the lowest available keypoint number.

Type

Type of input for *VALUE*.

RATIO

Value is the ratio of the distances between keypoints as follows: (*KP1*-*KPNEW*)/(*KP1*-*KP2*).

DIST

Value is the absolute distance between *KP1* and *KPNEW* (valid only if current coordinate system is Cartesian).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VALUE

Location of new keypoint, as defined by *Type* (defaults to 0.5). If *VALUE* is a ratio (*Type* = RATIO) and is less than 0 or greater than 1, the keypoint is created on the extended line. Similarly, if *VALUE* is a distance (*Type* = DIST) and is less than 0 or greater than the distance between *KP1* and *KP2*, the keypoint is created on the extended line.

Notes

Placement of the new keypoint depends on the currently active coordinate system (**CSYS (p. 441)**). If the coordinate system is Cartesian, the keypoint will lie on a straight line between *KP1* and *KP2*. If the system is not Cartesian (for example, cylindrical, spherical, etc.), the keypoint will be located as if on a line (which may not be straight) created in the current coordinate system between *KP1* and *KP2*. Note that solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>KP between KPs

---

KCENTER, *Type*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *KPNEW*

Creates a keypoint at the center of a circular arc defined by three locations.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Type of entity used to define the circular arc. The meaning of *VAL1* through *VAL4* will vary depending on *Type*. If *Type* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KP

Arc is defined by keypoints.

LINE

Arc is defined by locations on a line.

VAL1, VAL2, VAL3, VAL4

Values used to specify three locations on the arc (see table below).

KPNEW

Number assigned to new keypoint. Defaults to the lowest available keypoint number.

Definitions:

If *Type* = KP, inputs *VAL1* through *VAL4* are defined as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL1

First keypoint.

VAL2

Second keypoint.

VAL3

Third keypoint.

VAL4

Arc radius. If *VAL4* = 0 or blank (default), the arc is defined by the three keypoints specified as *VAL1*, *VAL2*, and *VAL3* and arc radius is not used. If *VAL4* is nonzero, *VAL1*, *VAL2*, and *VAL4* are used to calculate the center point, and *VAL3* is used to locate the center point as follows:

***VAL4* > 0**

Center point and *VAL3* are on the same side of the line between the first two keypoints.

***VAL4* < 0**

Center point and *VAL3* are on opposite sides of the line between the first two keypoints.

If *Type* = LINE, inputs *VAL1* through *VAL4* are defined as follows:

VAL1

Line number.

VAL2

Line ratio (0 to 1) indicating the first location (defaults to 0).

VAL3

Line ratio (0 to 1) indicating the second location (defaults to 0.5).

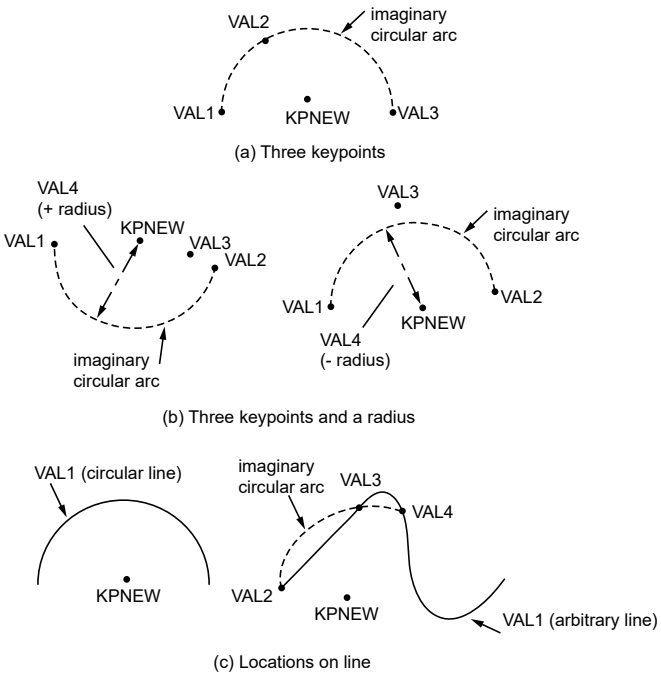
VAL4

Line ratio (0 to 1) indicating the third location (defaults to 1).

Notes

KCENTER (p. 973) should be used in the Cartesian coordinate system (**CSYS (p. 441)**,0) only. This command provides three methods to define a keypoint at the center of three locations. As shown below, the center point can be calculated based on a) three keypoints, b) three keypoints and a radius, or c) three locations on a line. Note that for method c, if a circular line is specified by *VAL1*, *VAL2* through *VAL4* are not needed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>3 keypoints

Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>3 KPs and radius

Main Menu>Preprocessor>Modeling>Create>Keypoints>KP at center>Location on line

---

KCLEAR, *NP1*, *NP2*, *NINC*

Deletes nodes and point elements associated with selected keypoints.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1, *NP2*, *NINC*

Delete mesh for keypoints *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the mesh for all selected keypoints (**KSEL (p. 995)**) is deleted. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1*.

Notes

Deletes *all* nodes and point elements associated with selected keypoints (regardless of whether the nodes or elements are selected). Nodes associated with non-point elements will not be deleted. Attributes assigned as a result of **KATT (p. 969)** are maintained. In the program's response to the command, if a keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Keypoints

KDELE, *NP1*, *NP2*, *NINC*

Deletes unmeshed keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1*, *NP2*, *NINC

Delete keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints (**KSEL (p. 995)**) are deleted. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

Notes

Deletes selected keypoints. A keypoint attached to a line cannot be deleted unless the line is first deleted.

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Keypoints

KDIST, *KP1*, *KP2*

Calculates and lists the distance between two keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KP1

First keypoint in distance calculation. If *KP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KP2

Second keypoint in distance calculation.

Notes

KDIST (p. 976) lists the distance between keypoints *KP1* and *KP2*, as well as the current coordinate system offsets from *KP1* to *KP2*, where the X, Y, and Z locations of *KP1* are subtracted from the X, Y, and Z locations of *KP2* (respectively) to determine the offsets. **KDIST (p. 976)** is valid in any coordinate system except toroidal (**CSYS (p. 441)**,3).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KDIST (p. 976) returns a variable, called "*_RETURN*," which contains the distance value. You can use this value for various purposes; for example, to set the default number of line divisions to be generated along region boundary lines (**ESIZE (p. 668)**,*_RETURN*). In interactive mode, you can access this command by using the Model Query Picker (**Utility Menu> List> Picked Entities**), where you can also access automatic annotation functions, and display the value on your model.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>KP distances

KEEP, *Key*

Stores POST26 definitions and data during active session.

POST26 (p. 58): Display (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

State or value

On or 1

Allows you to exit and reenter **/POST26 (p. 1462)** without losing your current time history variable information. Keeps a cache of the **/POST26 (p. 1462)** variable information including the active file name (**FILE (p. 752)**), variable definitions (**NSOL (p. 1281)**, **ESOL (p. 672)**, **RFORCE (p. 1607)**, and **SOLU (p. 1819)**) and stored variable data in memory for the current Mechanical APDL session.

Off or 0

/POST26 (p. 1462) variable information is deleted when you exit **/POST26 (p. 1462)**.

Command Default

ON - Hold time history information in memory. You can, for example, move back and forth between **/POST1 (p. 1461)** and **/POST26 (p. 1462)** without redefining and storing the time history variables each time you enter **/POST26 (p. 1462)**.

Notes

Your variable information is saved in memory only for the current active Mechanical APDL session. It is deleted when you exit Mechanical APDL. This information is also deleted when you issue **/CLEAR (p. 350)**, **RESUME (p. 1601)**, **SOLVE (p. 1822)**, or **RESET (p. 1598)**.

When you reenter **/POST26 (p. 1462)** all time history variable data is available for use. When you issue **STORE (p. 1867)**,*NEW*, variable definitions created by math operations such as **ADD (p. 95)** or **PROD (p. 1515)** will not be restored. However, variables defined with **NSOL (p. 1281)**, **ESOL (p. 672)**, **RFORCE (p. 1607)**, and **SOLU (p. 1819)** will be restored. Only the last active results file name is kept in memory (**FILE (p. 752)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Commands such as **LAYERP26 (p. 1016)**, **SHELL (p. 1784)**, and **FORCE (p. 772)** that specify the location or a component of data to be stored will retain the setting at the time of exiting **/POST26 (p. 1462)** .

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

KESIZE, *NPT*, *SIZE*, *FACT1*, *FACT2*

Specifies the edge lengths of the elements nearest a keypoint.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NPT

Number of the keypoint whose lines will be adjusted. If ALL, use all selected keypoints (**KSEL (p. 995)**). If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

SIZE

Size of elements along lines nearest keypoint *NPT* (overrides any other specified size). If *SIZE* is zero (or blank), use *FACT1* or *FACT2*.

FACT1

Scale factor applied to a previously defined *SIZE*. Not used if *SIZE* is input.

FACT2

Scale factor applied to the minimum element division at keypoint *NPT* for any attached line. This feature is useful with adaptive mesh refinement. Not used if *SIZE* or *FACT1* is input.

Notes

Affects only the line divisions adjacent to the keypoint on lines not previously assigned divisions by other line commands (**LESIZE (p. 1041)**, etc.). The remaining line divisions are determined from the division nearest the keypoint at the other end of the line (specified by another **KESIZE (p. 978)** command or the **ESIZE (p. 668)** command). Divisions are transferred to the lines during the mesh operation. If smart element sizing is being used (**SMRTSIZE (p. 1811)**), **KESIZE (p. 978)** specifications may be overridden (that is, a smaller element size may be used) to accommodate curvature and small features.

This command is valid in any processor. The command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>All KPs

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>Clr Size

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Keypoints>Picked KPs

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYOPT, *ITYPE*, *KNUM*, *VALUE*

Sets element key options.

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITYPE

Element type number as defined on the **ET (p. 686)** command.

The following labels are valid input for contact elements:

CONT

Set element key options for all contact element types, CONTA172, CONTA174, CONTA175, and CONTA177.

TARG

Set element key options for all target element types, TARGE169 and TARGE170.

GCN

Set element key options for all contact element types used in a general contact definition (that is, all contact elements having a real constant set number = 0). See Notes (p. 980) for additional information specific to general contact.

JOIN

Set element key options for all joint element types (MPC184 with KEYOPT(1) > 2). Valid for joint elements only. See Notes (p. 980) for additional information specific to joint elements.

KNUM

Number of the KEYOPT to be defined (KEYOPT(*KNUM*)).

VALUE

Value of this KEYOPT.

Notes

The **KEYOPT (p. 979)** command is an alternative method for inputting element key option (KEYOPT) values via the **ET (p. 686)** command. (Issue the **ET (p. 686)** command *first* to define *ITYPE*).

The **KEYOPT (p. 979)** command is required for inputting key options numbered higher than six (that is, > KEYOPT(6)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Special Usage for Structural Elements

If **ETCONTROL (p. 701)**,SET is enabled, key options that you specify via the **KEYOPT (p. 979)** command might be overridden for many structural elements. For more information, see Automatic Selection of Element Technologies and Formulations in the *Element Reference*.

Special Usage for General Contact

Specify *ITYPE* = GCN to set element key options for all contact element types used in a general contact definition. You can selectively set element key options for multiple contact element types in a general contact definition by setting *ITYPE* to a valid label (ALL_EDGE, ALL_FACE, ALL_VERT, ALL_TOP, or ALL_BOT) or by inputting a node component name with or without a component name extension (_EDGE, _FACE, _VERT, _TOP, or _BOT). For more information, see Defining Non-Default Contact Settings in the *Contact Technology Guide*.

Special Usage for Joint Elements

Specify *ITYPE* = JOIN to set element key options for all joint element types in the model. Use this option to switch between Lagrange Multiplier-based and penalty-based joint element formulations.

Menu Paths

Main Menu>Preprocessor>Element Type>Add/Edit/Delete

KEYPTS

Specifies "Keypoints" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEYW

Sets a keyword used by the GUI for context filtering (GUI).

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a program-generated command and is not intended for your use. It is included here in the documentation because it might appear in the log file (`Jobname.log`). When using log files, or portions of log files, as input, any included **KEYW (p. 981)** commands may be left as is, or removed, without consequence.

Menu Paths

Main Menu>Preferences

KFILL,*NP1*,*NP2*,*NFILL*,*NSTRT*,*NINC*,*SPACE*

Generates keypoints between two keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1, NP2

Beginning and ending keypoints for fill-in. *NP1* defaults to next to last keypoint specified, *NP2* defaults to last keypoint specified. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NFILL

Fill *NFILL* keypoints between *NP1* and *NP2* (defaults to |*NP2*-*NP1*|-1). *NFILL* must be positive.

NSTRT

Keypoint number assigned to first filled-in keypoint (defaults to *NP1* + *NINC*).

NINC

Add this increment to each of the remaining filled-in keypoint numbers (may be positive or negative). Defaults to (*NP2*-*NP1*)/(*NFILL* + 1), that is, linear interpolation.

SPACE

Spacing ratio. Ratio of last division size to first division size. If > 1.0, divisions increase. If < 1.0, divisions decrease. Ratio defaults to 1.0 (uniform spacing).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates keypoints (in the active coordinate system) between two existing keypoints. The two keypoints may have been defined in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Any number of keypoints may be filled in and any keypoint numbering sequence may be assigned.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>Fill between KPs

---

KGEN, *ITIME*, *NP1*, *NP2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional keypoints from a pattern of keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME

Do this generation operation a total of *ITIME* times, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be more than 1 for generation to occur.

NP1*, *NP2*, *NINC

Generate keypoints from the pattern of keypoints beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is all selected keypoints (**KSEL (p. 995)**). If *NP1* is negative, *NP2* and *NINC* are ignored and the last |*NP1*| keypoints (in sequence from the highest keypoint number) are used as the pattern to be repeated. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

DX*, *DY*, *DZ

Keypoint location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, Dθ, DΦ for spherical).

KINC

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and point elements associated with the original keypoints, if they exist.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Do not generate nodes and elements.

IMOVE

Specifies whether keypoints will be moved or newly defined:

0

Generate additional keypoints as requested with the *ITIME* argument.

1

Move original keypoints to new position retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates additional keypoints (and corresponding mesh) from a given keypoint pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Generation is done in the active coordinate system. Keypoints in the pattern may have been defined in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Keypoints

KL, *NL1*, *RATIO*, *NK1*

Generates a keypoint at a specified location on an existing line.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the line. If negative, the direction of line (as interpreted for *RATIO*) is reversed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

RATIO

Ratio of line length to locate keypoint. Must be between 0.0 and 1.0. Defaults to 0.5 (divide the line in half).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NK1

Number to be assigned to keypoint generated at division location (defaults to lowest available keypoint number (**NUMSTR (p. 1301)**)).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>On Line

Main Menu>Preprocessor>Modeling>Create>Keypoints>On Line w/Ratio

---

KLIST, *NP1*, *NP2*, *NINC*, *Lab*

Lists the defined keypoints or hard points.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1*, *NP2*, *NINC

List keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL (default), *NP2* and *NINC* are ignored and all selected keypoints (**KSEL (p. 995)**) are listed. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

Lab

Coordinate listing key:

(blank)

List all keypoint information.

COORD

Suppress all but the keypoint coordinates (shown to a higher degree of accuracy than when displayed with all information).

HPT

List only hard point information.

Notes

Lists keypoints in the active display coordinate system (**DSYS (p. 582)**). An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **KATT (p. 969)** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer (**TYPE (p. 2036)**, **MAT (p. 1111)**, **REAL (p. 1579)**, or **ESYS (p. 685)**) that was active during meshing (and will be reset to zero if the mesh is cleared).

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>List>Keypoints>Coordinates +Attributes

Utility Menu>List>Keypoints>Coordinates only

Utility Menu>List>Keypoints>Hard Points

KMESH, *NP1*, *NP2*, *NINC*

Generates nodes and point elements at keypoints.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1*, *NP2*, *NINC

Mesh keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints (**KSEL (p. 995)**) are meshed. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1*.

Notes

Missing nodes required for the generated elements are created and assigned the lowest available numbers.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Keypoints

KMODIF, *NPT*, *X*, *Y*, *Z*

Modifies an existing keypoint.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NPT

Modify coordinates of this keypoint. If *NPT* = ALL, modify coordinates of all selected keypoints (**KSEL (p. 995)**). If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

X*, *Y*, *Z

Replace the previous coordinate values assigned to this keypoint with these corresponding coordinate values. Values are interpreted according to the active coordinate system (R, θ, Z for cylindrical, R, θ,Φ for spherical). If *X* = P, graphical picking is used to locate keypoint and *Y* and *Z* are ignored. A blank retains the previous value. You cannot specify *Y* = P.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Lines, areas, and volumes attached to the modified keypoint (if any) must all be selected and will be redefined using the active coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Caution:

Redefined entities may be removed from any defined components and assemblies. Nodes and elements will be automatically cleared from any redefined keypoints, lines, areas, or volumes.

The **KMODIF (p. 985)** command moves keypoints for geometry modification without validating underlying entities. To merge keypoints and update higher order entities, issue the **NUMMRG (p. 1295)** command instead.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>Set of KPs

Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>Single KP

KMOVE, *NPT*, *KC1*, *X1*, *Y1*, *Z1*, *KC2*, *X2*, *Y2*, *Z2*

Calculates and moves a keypoint to an intersection.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NPT

Move this keypoint. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

KC1

First coordinate system number. Defaults to 0 (global Cartesian).

X1, Y1, Z1

Input one or two values defining the location of the keypoint in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R1, θ1, Z1 for cylindrical, or R1, θ1, φ1 for spherical.

KC2

Second coordinate system number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X2, Y2, Z2

Input two or one value(s) defining the location of the keypoint in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Arguments are R2, θ2, Z2 for cylindrical, or R2, θ2, φ2 for spherical.

Notes

Calculates and moves a keypoint to an intersection location. The keypoint must have been previously defined (at an approximate location) or left undefined (in which case it is internally defined at the **SOURCE (p. 1824)** location). The actual location is calculated from the intersection of three surfaces (implied from three coordinate constants in two different coordinate systems). Note that solid modeling in a toroidal coordinate system is not recommended. See the **MOVE (p. 1158)** command for surface and intersection details. The three (of six) constants easiest to define should be used. The program will calculate the remaining three coordinate constants. All arguments, except *KC1*, must be input. Use the repeat command (***REPEAT (p. 1586)**) after the **KMOVE (p. 986)** command to move a series of keypoints, if desired.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Keypoints>To Intersect

---

KNODE, *NPT*, *NODE*

Defines a keypoint at an existing node location.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NPT

Arbitrary reference number for keypoint. If zero, the lowest available number is assigned (**NUMSTR (p. 1301)**).

NODE

Node number defining global X, Y, Z keypoint location. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Keypoints>On Node

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KPLOT, *NP1*, *NP2*, *NINC*, *Lab*

Displays the selected keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1*, *NP2*, *NINC

Display keypoints from *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL (default), *NP2* and *NINC* are ignored and all selected keypoints (**KSEL (p. 995)**) are displayed.

Lab

Determines what keypoints are plotted (one of the following):

(blank)

Plots all keypoints.

HPT

Plots only those keypoints that are hard points.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Keypoints>Hardpoints

Utility Menu>Plot>Keypoints>Keypoints

Utility Menu>Plot>Specified Entities>Keypoints

KPSCALE, *NP1*, *NP2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of (meshed) keypoints from a pattern of keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1*, *NP2*, *NINC

Set of keypoints (*NP1* to *NP2* in steps of *NINC*) that defines the pattern to be scaled. *NP2* defaults to *NP1*, *NINC* defaults to 1. If *NP1* = ALL, *NP2* and *NINC* are ignored and the pattern is defined by all selected keypoints. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RX, RY, RZ

Scale factors to be applied to the X, Y, Z keypoint coordinates in the active coordinate system (RR, Rθ, RZ for cylindrical; RR, Rθ, RΦ for spherical). The Rθ and RΦ scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, an *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints.) Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC

Increment to be applied to the keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Nodes and point elements associated with the original keypoints will be generated (scaled) if they exist.

1

Nodes and point elements will *not* be generated.

IMOVE

Specifies whether keypoints will be moved or newly defined:

0

Additional keypoints will be generated.

1

Original keypoints will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a scaled set of keypoints (and corresponding mesh) from a pattern of keypoints. The MAT, TYPE, REAL, and ESYS attributes are based on the keypoints in the pattern and not the current settings. Scaling is done in the active coordinate system. Keypoints in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KREFINE, *NP1*, *NP2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified keypoints.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NP1, NP2, NINC

Keypoints (*NP1* to *NP2* in increments of *NINC*) around which the mesh is to be refined. *NP2* defaults to *NP1*, and *NINC* defaults to 1. If *NP1* = ALL, *NP2* and *NINC* are ignored and all selected keypoints are used for refinement. If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH

Depth of mesh refinement in terms of the number of elements outward from the indicated keypoints (defaults to 1).

POST

Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF

No postprocessing will be done.

SMOOTH

Smoothing will be done. Node locations may change.

CLEAN

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (Mechanical APDL ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

KREFINE (p. 990) performs local mesh refinement around the specified keypoints. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

KREFINE (p. 990) refines all area elements and tetrahedral volume elements that are adjacent to the specified keypoints. Any volume elements that are adjacent to the specified keypoints, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes (**IC (p. 921)**), coupled nodes (**CP (p. 421)** family of commands), constraint equations (**CE (p. 297)** family of commands), or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. See Revising Your Model in the *Modeling and Meshing Guide* for additional restrictions on mesh refinement.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Keypoints

---

*KRON, *M1*, *M2*, *M3*

Computes the Kronecker product of two matrices/vectors, $M_3 = M_1 \otimes M_2$.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

M1

Name of matrix (or vector) M_1. Must have been previously specified (***DMAT (p. 551)**, ***SMAT (p. 1801)**, ***VEC (p. 2085)**, etc.)

M2

Name of matrix (or vector) M_2. Must have been previously specified (***DMAT (p. 551)**, ***SMAT (p. 1801)**, ***VEC (p. 2085)**, etc.)

M3

Name (up to 32 characters) of resulting matrix (or vector) M_3. Must be specified (no default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Details of the operation are demonstrated in the example below:

$$\begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \otimes \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix} = \begin{bmatrix} a_{11}\times b_{11} & a_{11}\times b_{12} & a_{12}\times b_{11} & a_{12}\times b_{12} \\ a_{11}\times b_{21} & a_{11}\times b_{22} & a_{12}\times b_{21} & a_{12}\times b_{22} \\ a_{21}\times b_{11} & a_{21}\times b_{12} & a_{22}\times b_{11} & a_{22}\times b_{12} \\ a_{21}\times b_{21} & a_{21}\times b_{22} & a_{22}\times b_{21} & a_{22}\times b_{22} \end{bmatrix}$$

Input matrices (or vectors) can be any of the supported types: integer, double precision real, or complex double precision.

Input matrices can be sparse or dense, and their dimensions do not have to be the same.

Output matrices are always dense, regardless of the input matrices type.

Since the ***KRON (p. 991)** command is manipulating dense matrices, it should only be used on small or medium-size matrices for performances reasons.

Menu Paths

This command cannot be accessed from a menu.

KRYOPT, *MAXDIM*, --, --, *RESTOL*, *OrthTol*

Specifies solution options for a Krylov method harmonic analysis.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAXDIM

Maximum dimension of subspace. The default size is automatically determined by the program (ranges around 50).

--, --

Unused fields.

RESTOL

Tolerance used to verify the L-2 norm values of calculated residuals and issue a warning if *RESTOL* is exceeded throughout the entire frequency range. Default = 0.05.

OrthTol

Tolerance that controls the orthonormality check of the Krylov subspace vectors. Valid values range from 0 to 1. The default is 1E-8.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

The default values listed for each argument above are used.

Notes

This command is used to specify solution options for a harmonic analysis solved with the Krylov method (*Method* = KRYLOV on **HROPT (p. 912)**, see also Frequency-Sweep Harmonic Analysis via the Krylov Method in the *Structural Analysis Guide*).

Increasing subspace size (*MAXDIM*) improves solution accuracy with the trade-off of increased computational cost and additional memory requirements.

The *RESTOL* and *OrthTol* options do not affect the accuracy of the solution. They simply control the output of messages displayed regarding the accuracy of the final computed solution or orthonormality of the computed subspace.

The program computes the condition number of a matrix created based on the dot products of the Krylov subspace vectors with each other. When the condition number of this matrix is closer to 1, it means the subspace vectors are orthonormal. When it is closer to 0, that means the subspace vectors are not orthonormal. The *OrthTol* value is used in the following formula to determine the orthonormality of the computed Krylov subspace: $1-\frac{1}{\text{Condition Number}}<\text{OrthTol}$. When this condition is true, the Krylov subspace is determined to be sufficiently orthonormal, and vice versa.

Menu Paths

This command cannot be accessed from a menu.

KSCALE, *KINC*, *NP1*, *NP2*, *NINC*, *RX*, *RY*, *RZ*

Generates a scaled pattern of keypoints from a given keypoint pattern.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KINC

Do this scaling operation one time, incrementing all keypoints in the given pattern by *KINC*. If *KINC* = 0, keypoints will be redefined at the scaled locations.

NP1*, *NP2*, *NINC

Scale keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints (**KSEL (p. 995)**). If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RX, RY, RZ

Scale factor ratios. Scaling is relative to the origin of the active coordinate system (RR, Rθ, RZ for cylindrical, RR, Rθ, RΦ for spherical). If > 1.0, pattern is enlarged. If < 1.0, pattern is reduced. Ratios each default to 1.0.

Notes

Generates a scaled pattern of keypoints from a given keypoint pattern. Scaling is done in the active coordinate system (see analogous node scaling (**NSCALE (p. 1265)**)). Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

This command cannot be accessed from a menu.

KSCON, *NPT*, *DELR*, *KCTIP*, *NTHET*, *RRAT*

Specifies a keypoint about which an area mesh will be skewed.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NPT

Keypoint number at concentration. If *NPT* = ALL, use all selected keypoints. If remaining fields are blank, remove concentration from this keypoint (if unmeshed). If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

DELR

Radius of first row of elements about keypoint.

KCTIP

Crack tip singularity key:

0

Do not skew midside nodes, if any, within the element.

1

Skew midside nodes of the first row of elements to the 1/4 point for crack tip singularity.

NTHET

Number of elements in circumferential direction (defaults to approximately one per 45° (or one per 30°, if *KCTIP* = 1)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RRAT

Ratio of 2nd row element size to *DELR* (defaults to 0.75, or 0.5 if *KCTIP* = 1).

Notes

Defines a concentration keypoint about which an area mesh will be skewed. Useful for modeling stress concentrations and crack tips. During meshing, elements are initially generated circumferentially about, and radially away, from the keypoint. Lines attached to the keypoint are given appropriate divisions and spacing ratios. Only one concentration keypoint per unmeshed area is allowed. Use **KSCON (p. 994)**,STAT to list current status of concentration keypoints. The **KSCON (p. 994)** command does not support 3D modeling.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>Concentrat KPs>Create

Main Menu>Preprocessor>Meshing>Size Cntrls>Concentrat KPs>List

---

KSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Selects a subset of keypoints or hard points.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

ALL

Restore the full set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item

Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to KP.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

VMIN

Minimum value of item range. Ranges are keypoint numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM (p. 356)** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **KLIST (p. 984)** command for a discussion of signed attributes.

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN*.

VINC

Value increment within range. Used only with integer ranges (such as for keypoint numbers). Defaults to 1. *VINC* cannot be negative.

KABS

Absolute value key:

0

Check sign of value during selection.

1

Use absolute value during selection (sign ignored).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

All keypoints are selected.

Notes

Selects a subset of keypoints or hard points. For example, to select a new set of keypoints based on keypoint numbers 1 through 7, use **KSEL (p. 995)**,S,KP,,1,7. The selected subset is used when the ALL label is entered (or implied) on other commands, such as **KLIST (p. 984)**,ALL. Only data identified by keypoint number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

This command is valid in any processor.

For selections based on non-integer numbers (coordinates, results, etc.), items that are within the range *VMIN* -*Toler* and *VMAX* + *Toler* are selected. The default tolerance *Toler* is based on the relative values of *VMIN* and *VMAX* as follows:

- If *VMIN* = *VMAX*, *Toler* = 0.005 x *VMIN*.
- If *VMIN* = *VMAX* = 0.0, *Toler* = 1.0E-6.
- If *VMAX* ≠ *VMIN*, *Toler* = 1.0E-8 x (*VMAX* - *VMIN*).

Use the **SELTOL (p. 1710)** (p. 1710) command to override this default and specify *Toler* explicitly.

Table 192: KSEL - Valid Item and Component Labels

Valid Item and Component Labels KSEL (p. 995), *Type, Item, Comp, VMIN, VMAX, VINC, KABS*

Item	Comp	Description
KP		Keypoint number.
EXT		Keypoint numbers on exterior of selected lines (ignore remaining fields).
HPT		Hard point number.
LOC	X,Y,Z	X, Y, or Z location in the active coordinate system.
MAT		Material number associated with the keypoint.
TYPE		Element type number associated with the keypoint.
REAL		Real constant set number associated with the keypoint.
ESYS		Element coordinate system associated with the keypoint.

Menu Paths

Utility Menu>Select>Entities

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KSLL, *Type*

Selects those keypoints contained in the selected lines.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of keypoint select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

KSLN, *Type*

Selects those keypoints associated with the selected nodes.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of keypoint select:

S

Select a new set (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Notes

Valid only if the nodes were generated by a *meshing* operation (**KMESH (p. 985)**, **LMESH (p. 1057)**, **AMESH (p. 121)**, **VMESH (p. 2140)**) on a solid model that contains the associated keypoints.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

KSUM

Calculates and prints geometry statistics of the selected keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Calculates and prints geometry statistics (centroid location, moments of inertia, etc.) associated with the selected keypoints. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed, irrespective of any material associations (**KATT (p. 969)**, **MAT (p. 1111)**). Items calculated by **KSUM (p. 999)** and later retrieved by a ***GET (p. 797)** or ***VGET (p. 2118)** command are valid only if the model is not modified after the **KSUM (p. 999)** command is issued.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KSYMM, *Ncomp*, *NP1*, *NP2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates a reflected set of keypoints.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ncomp

Symmetry key:

X

X (or R) symmetry (default).

Y

Y (or θ) symmetry.

Z

Z (or Φ) symmetry.

NP1*, *NP2*, *NINC

Reflect keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints (**KSEL (p. 995)**). If *Ncomp* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Generate nodes and point elements associated with the original keypoints, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether keypoints will be moved or newly defined:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Generate additional keypoints.

1

Move original keypoints to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of keypoints (and corresponding mesh) from a given keypoint pattern by a symmetry reflection (see analogous node symmetry command, **NSYM (p. 1289)**). The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. Keypoints in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Keypoints

KTRAN, *KCNTO*, *NP1*, *NP2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of keypoints to another coordinate system.

PREP7 (p. 22): Keypoints (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCNTO

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system.

NP1*, *NP2*, *NINC

Transfer keypoints from pattern beginning with *NP1* to *NP2* (defaults to *NP1*) in steps of *NINC* (defaults to 1). If *NP1* = ALL, *NP2* and *NINC* are ignored and pattern is all selected keypoints (**KSEL (p. 995)**). If *NP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NP1* (*NP2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Generate nodes and point elements associated with the original keypoints, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether keypoints will be moved or newly defined:

0

Generate additional keypoints.

1

Move original keypoints to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old keypoints are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of keypoints (and corresponding mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER (p. 2011)**). The MAT, TYPE, REAL, and ESYS attributes are based upon the keypoints in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. Coordinate values are interpreted in the active coordinate system and are transferred directly. Solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Keypoints

---

KUSE, *KEY*

Specifies whether or not to reuse factorized matrices.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Reuse key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Program decides whether or not to reuse the previous factorized matrices.

1

Force the previous factorized matrices to be reused. Used mainly in a restart. Forcing reuse of the matrices is a nonstandard use of the program and should be done with caution. For instance, using this option and changing the number of elements, or the number or type of degrees of freedom, may cause an abort.

-1

All element matrices are reformed and are used to reform new factorized matrices.

Command Default

Program makes decision.

Notes

Overrides the program logic to determine whether or not to reuse the previous factorized matrices for each substep of this load step. Applies only to static or full transient analyses. For more details see Other General Options in the *Basic Analysis Guide*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Reuse Factorized Matrix

Main Menu>Solution>Load Step Opts>Other>Reuse Factorized Matrix

---

KWPAVE, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*, *P9*

Moves the working plane origin to the average location of keypoints.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1, P2, P3, . . . , P9

Keypoints used in calculation of the average. At least one must be defined. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Moves the origin of the working plane to the average of the specified keypoints. Averaging is based on the active coordinate system.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>WorkPlane>Offset WP to>Keypoints

KWPLAN, *WN*, *KORIG*, *KXAX*, *KPLAN*

Defines the working plane using three keypoints.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

KORIG

Keypoint number defining the origin of the working plane coordinate system. If *KORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

KXAX

Keypoint number defining the x-axis orientation (defaults to the x-axis being parallel to the global X-axis; or if the normal to the working plane is parallel to the global X-axis, then defaults to being parallel to the global Y-axis).

KPLAN

Keypoint number defining the working plane (the normal defaults to the present display view (**/VIEW (p. 2131)**) of window *WN*).

Notes

Defines a working plane to assist in picking operations using three keypoints as an alternate to the **WPLANE (p. 2180)** command. The three keypoints also define the working plane coordinate system. A minimum of one keypoint (at the working plane origin) is required. Immediate mode may also be active. See **WPSTYL (p. 2182)** command to set the style of working plane display.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Align WP with>Keypoints

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

L Commands

L, *P1*, *P2*, *NDIV*, *SPACE*, *XV1*, *YV1*, *ZV1*, *XV2*, *YV2*, *ZV2*

Defines a line between two keypoints.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1

Keypoint at the beginning of line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2

Keypoint at the end of line.

NDIV

Number of element divisions within this line. Normally this field is not used; specifying divisions with **LESIZE (p. 1041)**, etc. is recommended.

SPACE

Spacing ratio. Normally this field is not used, as specifying spacing ratios with the **LESIZE (p. 1041)** command is recommended. If positive, *SPACE* is the nominal ratio of the last division size (at *P2*) to the first division size (at *P1*). If the ratio is greater than 1, the division sizes increase from *P1* to *P2*, and if less than 1, they decrease. If *SPACE* is negative, then |*SPACE*| is the nominal ratio of the center division size to those at the ends.

The following fields are used only if specified end slopes on the line are desired, otherwise zero curvature end slopes will be automatically calculated to produce a line which is "straight" in the active coordinate system. To specify end slopes, use the following fields to define a "slope vector" (one for each end of the line, if desired) that has its tail at the origin and its head at the point *XV*, *YV*, *ZV* in the active coordinate system (**CSYS (p. 441)**). The corresponding end slope of the line will then be parallel to this "slope vector."

XV1, YV1, ZV1

Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P1* end of the line. The tail of the vector is at the origin of the coordinate system.

XV2, YV2, ZV2

Location of the head of the "slope vector" corresponding to the slope at the *P2* end of the line.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines a line between two keypoints from *P1* to *P2*. The line shape may be generated as "straight" (in the active coordinate system) or curved. The line shape is invariant with coordinate system after it is generated. Note that solid modeling in a toroidal coordinate system is not recommended. A curved line is limited to 180°. Lines may be redefined only if not yet attached to an area.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>In Active Coord

---

L2ANG, *NL1*, *NL2*, *ANG1*, *ANG2*, *PHIT1*, *PHIT2*

Generates a line at an angle with two existing lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the first line to be hit (touched by the end of the new line). If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NL2

Number of the second line to be hit. If negative, assume *P3* is the second keypoint of the line instead of the first.

ANG1

Angle of intersection (usually zero or 180) of generated line with tangent to first line.

ANG2

Angle of intersection (usually zero or 180) of generated line with tangent to second line.

PHIT1

Number to be assigned to keypoint generated at hit location on first line (defaults to lowest available keypoint number (**NUMSTR (p. 1301)**)).

PHIT2

Number to be assigned to keypoint generated at hit location on second line (defaults to lowest available keypoint number (**NUMSTR (p. 1301)**)).

Notes

Generates a straight line (*PHIT1-PHIT2*) at an angle (*ANG1*) with an existing line *NL1* (*P1-P2*) and which is also at an angle (*ANG2*) with another existing line *NL2* (*P3-P4*). If the angles are zero the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

generated line is tangent to the two lines. The *PHIT1* and *PHIT2* locations on the lines are automatically calculated. Line *P1-P2* becomes *P1-PHIT1*, *P3-P4* becomes *P3-PHIT2*, and new lines *PHIT1-P2*, *PHIT2-P4*, and *PHIT1-PHIT2* are generated. Line divisions are set to zero (use **LESIZE (p. 1041)**, etc. to modify).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Angle to 2 Lines

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Norm to 2 Lines

---

L2TAN, *NL1*, *NL2*

Generates a line tangent to two lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the first line generated line is tangent to. If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NL2

Number of the second line generated line is tangent to. If negative, assume *P3* is the second keypoint of the line instead of the first.

Notes

Generates a line (*P2-P3*) tangent at point *P2* to line *NL1* (*P1-P2*) and tangent at point *P3* to line *NL2* (*P3-P4*).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Tan to 2 Lines

---

LANBOPTION, *StrmCk*, *--*, *AltMeth*, *Memory_Option*

Specifies Block Lanczos eigensolver options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

StrmCk

Controls whether the Block Lanczos eigensolver will perform a Sturm sequence check:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Do not perform the Sturm sequence check (default).

ON

Perform a Sturm sequence check. This requires additional matrix factorization (which can be expensive), but does help ensure that no modes are missed in the specified range.

--

Unused field.

AltMeth

ALT1

Alternative version of the Block Lanczos eigensolver for more difficult modal or buckling problems. This version of Block Lanczos only runs in shared-memory parallel (SMP) mode. If the analysis is run in distributed-memory parallel (DMP) mode, it will switch to SMP mode when this Alternative Block Lanczos solver is invoked, and resume in DMP mode after the eigensolution.

Memory_Option

Memory allocation option:

DEFAULT

Default memory configuration (default). Everything is determined dynamically with respect to current machine resources.

INCORE

Fully in-core memory configuration.

MIX1

First level of mixed in-core / out-of-core configuration.

MIX2

Second level of mixed in-core / out-of-core configuration.

OUTOFCORE

Fully out-of-core memory configuration.

Command Default

The Sturm sequence check is off for the Block Lanczos eigensolver when it is used in a modal analysis, and on when it is used in a buckling analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default memory allocation strategy is used. Mechanical APDL evaluates the resources of the machine to choose the in-core or out-of-core mode.

Notes

LANBOPTION (p. 1007) specifies options to be used with the Block Lanczos eigensolver during an eigenvalue buckling analysis (**BUCOPT (p. 280)**,LANB) or a modal analysis (**MODOPT (p. 1140)**,LANB).

For more difficult eigenproblems, *AltMeth* = ALT1 could achieve better converged eigensolutions at the cost of more computing time. This ALT1 option is useful for double-checking solution accuracy. It should be used for difficult eigenproblems like those with many duplicated eigenmodes, or eigenbuckling problems with very thin beam/shell structures.

Memory Allocation Option

The Block Lanczos eigensolver algorithm allocates two main pools of memory:

1. Memory for the internal sparse solver iterations.
2. Memory for the specific Lanczos working arrays.

The following table shows how memory is allocated for each option.

Memory_Option	Sparse Solver	Working Arrays
INCORE	In-Core	In-Core
MIX1	In-Core	Out-of-Core
MIX2	Out-of-Core	In-Core
OUTOFCORE	Out-of-Core	Out-of-Core

The MIX1 configuration typically uses more memory than the MIX2 configuration, except when a large number of modes are requested for a small model.

Menu Paths

This command cannot be accessed from a menu.

LANG, *NL1*, *P3*, *ANG*, *PHIT*, *LOCAT*

Generates a straight line at an angle with a line.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the line to be hit (touched by the end of the new line). If negative, assume *P1* (see below) is the second keypoint of the line instead of the first. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P3

Keypoint at which generated line must end.

ANG

Angle of intersection of generated line *PHIT-P3* with tangent to line *P1-P2* at *PHIT*. If 0 (default), the generated line is tangent to *NL1* toward end P1; if 90, the generated line is perpendicular to *NL1*. If 180, the generated line is tangent to NL1 toward end P2. *ANG* can be any value, but is adjusted to the corresponding acute angle with respect to *LOCAT*. See "Notes" (p. 1010) for a discussion of accuracy.

PHIT

Number to be assigned to keypoint generated at hit location (defaults to lowest available keypoint number (**NUMSTR (p. 1301)**)).

LOCAT

Approximate location of *PHIT* in terms of the ratio of the distance along the line (*NL1*) to the length of the line. *LOCAT* can range from 0 to 1. If *LOCAT* is blank, the point will be located with less speed and accuracy, and an arbitrary location may result.

Notes

Generates a straight line (*PHIT-P3*) at an angle (*ANG*) with a line *NL1* (*P1-P2*). The location of *PHIT* on the line is automatically calculated. Line *P1-P2* becomes *P1-PHIT* and new lines *PHIT-P2* and *PHIT-P3* are generated. Line divisions are set to zero (use **LESIZE (p. 1041)**, etc. to modify).

PHIT is positioned closest to *LOCAT* for the given angle, *ANG*. To ensure better performance, it is recommended that *LOCAT* be input, even if it is 0.

The program uses an iterative procedure to position *PHIT*. The procedure is not exact, with the result that the actual value of *ANG* will sometimes differ slightly from the specified value.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>At angle to line

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Normal to Line

---

LARC, *P1*, *P2*, *PC*, *RAD*

Defines a circular arc.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1

Keypoint at one end of circular arc line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P2

Keypoint at other end of circular arc line.

PC

Keypoint defining plane of arc and center of curvature side (with positive radius). Must not lie along the straight line from *P1* to *P2*. *PC* need not be at the center of curvature.

RAD

Radius of curvature of the arc. If negative, assume center of curvature side is opposite to that defined by *PC*. If *RAD* is blank, *RAD* will be calculated from a curve fit through *P1*, *PC*, and *P2*.

Notes

Defines a circular arc line from *P1* to *P2*. The line shape is generated as circular, regardless of the active coordinate system. The line shape is invariant with coordinate system after it is generated.

When dealing with a large radius arc (1e3), or if the location of the arc you create is far away from the origin of your coordinate system, anomalies may occur. You can prevent this by creating the arc at a smaller scale, and then scaling the model back to full size (**LSSCALE (p. 1084)**).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>By End KPs & Rad

Main Menu>Preprocessor>Modeling>Create>Lines>Arcs>Through 3 KPs

---

/**LARC**, *XCENTR*, *YCENTR*, *XLRAD*, *ANGLE1*, *ANGLE2*

Creates annotation arcs (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCENTR

Arc X center location (-1.0 < X < 1.0).

YCENTR

Arc Y center location (-1.0 < Y < 1.0).

XLRAD

Arc radius length.

ANGLE1

Starting angle of arc.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANGLE2

Ending angle of arc. The arc is drawn counterclockwise from the starting angle, *ANGLE1*, to the ending angle, *ANGLE2*.

Notes

This command defines annotation arcs to be written directly onto the display at a specified location.

The command is generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.log`) if annotation is used. It is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

All arcs are shown on subsequent displays unless the annotation is turned off or deleted. Issueu **/LSPEC (p. 1081)** to set the attributes of the arc.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

LAREA, *P1*, *P2*, *NAREA*

Generates the shortest line between two keypoints on an area.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1

First keypoint of line to be generated. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2

Second keypoint of line to be generated.

NAREA

Area containing *P1* and *P2*, or area to which generated line is to be parallel.

Notes

Generates the shortest line between two keypoints, *P1* and *P2*, both of which lie on an area. The generated line will also lie on the area. *P1* and *P2* may also be equidistant (in global Cartesian space) from the area (and on the same side of the area), in which case a line parallel to the area is generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Overlaid on Area

---

LARGE, *IR*, *IA*, *IB*, *IC*, *Name*, - -, - -, *FACTA*, *FACTB*, *FACTC*

Finds the largest (the envelope) of three variables.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, IB, IC

Reference numbers of the three variables to be operated on. If only two, leave *IC* blank. If only one, leave *IB* blank also.

Name

Thirty-two character name for identifying the variable on the printout and displays. Embedded blanks are compressed upon output.

--, --

Unused fields.

FACTA, FACTB, FACTC

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Creates a new variable by finding the largest of up to three variables according to the operation:

IR = Largest of (*FACTA* x *IA*, *FACTB* x *IB*, *FACTC* x *IC*)

The comparison is done at each time location, so that the new variable is the "envelope" of the three existing variables.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Find Maximum

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LATT, *MAT*, *REAL*, *TYPE*, *- -*, *KB*, *KE*, *SECNUM*

Associates element attributes with the selected, unmeshed lines.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT, REAL, TYPE

Material number, real constant set number, and type number to be associated with selected, unmeshed lines.

- -

Unused field.

KB, KE

Beginning and ending orientation keypoints to be associated with selected, unmeshed lines. Mechanical APDL uses the location of these keypoints to determine how to orient beam cross sections during beam meshing. Beam elements may be created along a line with a constant orientation by specifying only one orientation keypoint (*KB*), or a pre-twisted beam may be created by selecting different orientation keypoints at each end of the line (*KB* and *KE*). (For a line bounded by two keypoints (*KP1* and *KP2*), the orientation vector at the beginning of the line extends from *KP1* to *KB*, and the orientation vector at the end of the line extends from *KP2* to *KE*. The orientation vectors are used to compute the orientation nodes of the elements.)

SECNUM

Section identifier to be associated with selected, unmeshed lines. For details, see the description of the **SECTYPE (p. 1697)** and **SECNUM (p. 1685)** commands.

Notes

The element attributes specified by the **LATT (p. 1014)** command will be used when the lines are meshed.

Lines subsequently generated from the lines will also have the attributes specified by *MAT*, *REAL*, *TYPE*, and *SECNUM*. If a line does not have these attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT (p. 1111)**, **REAL (p. 1579)**, **TYPE (p. 2036)**, and **SECNUM (p. 1685)** command settings.

In contrast, the values specified by *KB* and *KE* apply only to the selected lines; that is, lines subsequently generated from these lines will not share these attributes. Similarly, if a line does not have *KB* and *KE* attributes associated with it via the **LATT (p. 1014)** command at the time it is meshed, Mechanical APDL cannot obtain the attributes from elsewhere. See the discussion on beam meshing in Meshing Your Solid Model in the *Modeling and Meshing Guide* for more information.

Reissue the **LATT (p. 1014)** command (before lines are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association. If any of the arguments are defined as -1, then that value will be left unchanged in the selected set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In some cases, Mechanical APDL can proceed with a line meshing operation even when no logical element type has been assigned via **LATT (p. 1014)**,,,*TYPE* or **TYPE (p. 2036)**. See Meshing Your Solid Model in the *Modeling and Meshing Guide* for more information about setting element attributes.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Lines

Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Lines

LAYER, *NUM*

Specifies the element layer for which data are to be processed.

POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUM

Layer-processing mode:

N

The layer number to process. The default value is 0.

FCMAX

Processes the layer with the largest failure criteria.

Command Default

The default is *N* = 0, meaning that the entire element is considered to be the default "layer." Accordingly, the results data are from the bottom of the bottom layer and the top of the top layer.

Notes

Specifies the element layer for which results data are to be listed, plotted, or otherwise processed.

Applies to stress and strain data for layered elements SHELL181, SHELL281, ELBOW290, SOLID185, SOLID186, SOLSH190, SHELL208, SHELL209, REINF264, and REINF265; heat flux and heat gradient for SHELL131 and SHELL132.

For reinforcing elements, *N* is a given reinforcing member (individual reinforcing). Specifying *N* = 0 (default) or *N* = 1 selects the first reinforcing member.

The **SHELL (p. 1784)** command can then be used (with shell elements) to specify a location (TOP, MID, BOT) within the layer for output. (The **SHELL (p. 1784)** command does not apply to thermal shell elements SHELL131 and SHELL132.) Transverse shear stresses for MID are linearly averaged from TOP and BOT, and do not reflect a parabolic distribution. Setting KEYOPT(8) = 2 for SHELL181, SHELL281, SHELL208, SHELL209, and ELBOW290 writes the mid-surface values directly to the results file and yields more accurate values than linear averaging.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Because energy is a per-element quantity, you cannot use this command for energy output.

When using the **LAYER (p. 1015)** command with SHELL181, SOLID185, SOLID186, SOLSH190, SHELL208, SHELL209, SHELL281, and ELBOW290, KEYOPT(8) must be set to 1 (or 2 for SHELL181, SHELL281, ELBOW290, SHELL208, and SHELL209) in order to store results for all layers.

When *NUM* = FCMAX, you must provide the failure criterion input. If specifying input via the **FC (p. 732)** command, all structural elements are processed. For more information, see the documentation for the **FC (p. 732)** command.

Use this command with **RSYS (p. 1639)**,LSYS to display results in the layer coordinate system for a particular layer.

Menu Paths

Main Menu>General Postproc>Options for Outp

Utility Menu>List>Results>Options

LAYERP26, *NUM*

Specifies the element layer for which data are to be stored.

POST26 (p. 58): Controls (p. 59)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUM

Layer-processing mode:

N

The layer number to process. The default value is 1.

Command Default

The default is *N* = 1, meaning that results data are from the "first" layer. If KEYOPT(8) = 0, "first" layer actually means the bottom of the bottom layer and the top of the top layer.

Notes

Defines the element layer for which results data are to be stored for postprocessing. Applies to stress and strain data for layered elements SHELL181, SOLID185, SOLID186, SOLSH190, SHELL208, SHELL209, SHELL281, REINF265, and ELBOW290.

The **SHELL (p. 1784)** command can be used (for shell elements) to specify a location (TOP, MID, BOT) within the layer for selection on the **ESOL (p. 672)** command. Transverse shear stresses for MID are linearly averaged from TOP and BOT, and do not reflect a parabolic distribution. Setting KEYOPT(8) = 2 for SHELL181, SHELL208, SHELL209, SHELL281, and ELBOW290 writes the mid-surface values directly to the results file and yields more accurate values than linear averaging.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

That this command cannot be used for energy output, as energy is a per-element quantity.

When using the **LAYERP26 (p. 1016)** command with SHELL181, SOLID185, SOLID186, SOLSH190, SHELL208, or SHELL209, KEYOPT(8) must be set to 1 (or 2 for SHELL181, SHELL208, SHELL209, SHELL281, and ELBOW290) in order to store results for all layers.

In POST26, the **ESOL (p. 672)** data stored is based on the active **LAYERP26 (p. 1016)** specification at the time the data is stored. To store data at various specifications (for example, layers 2 and 5), issue a **STORE (p. 1867)** command before each new specification.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

LAYLIST, *IEL*, *LAYR1*, *LAYR2*, *Mplab1*, *Mplab2*

Lists real constants material properties for layered elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IEL

Element number to be listed. If ALL, list all selected elements (**ESEL (p. 661)**) of the appropriate type. If blank and the current element type is a layered element type, list data from the current real constant table in the layered format.

LAYR1, LAYR2

Range of layer numbers to be listed. If *LAYR1* is greater than *LAYR2*, a reverse order list is produced. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers if *LAYR1* is not input.

Mplab1, Mplab2

Material property labels (for example, EX) to be listed along with the layer real constants.

Notes

Lists real constants and any two material properties for layered shell and solid elements.

If matrix input is selected (KEYOPT(2) = 2 or 3), *LAYR1*, *LAYR2*, *Mplab1*, and *Mplab2* are not used.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Elements>Layered Elements

Utility Menu>List>Properties>Layer Data

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LAYPLOT, *IEL*, *LAYR1*, *LAYR2*

Displays the layer stacking sequence for layered elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IEL

Element number for the display. If blank and the current element type is a layered element type, display data from the current real constant table.

LAYR1, LAYR2

Range of layer numbers to be displayed. If *LAYR1* is greater than *LAYR2*, a reversed order display is produced. Up to 20 layers may be displayed at a time. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers (or to 19+*LAYR1*, if smaller) if *LAYR1* is not input.

Notes

Displays the layer-stacking sequence as defined in the real constant table for layered shell and solid elements in a form where the layers are visible (like a sheared deck of cards).

The element x-axis is shown as 0.0 degrees.

Layers are cross-hatched and color-coded for clarity. The hatch lines indicate the layer angle (real constant THETA) and the color coding is for material identification (real constant MAT).

The actual orientation of a specific layer in three-dimensional space can be seen using **/PSYMB (p. 1552)**,LAYR. To use **/PSYMB (p. 1552)**,LAYR with smeared reinforcing elements (REINF265), first set the vector-mode graphics option (**/DEVICE (p. 523)**,VECTOR,1).

Layer thickness can be displayed using the **/ESHAPE (p. 665)** and **EPLOT (p. 648)** commands.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Layered Elements

LCABS, *LCN0*, *KABS*

Specifies absolute values for load case operations.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCN0

Load case pointer number. If ALL, apply to all selected load cases (**LCSEL (p. 1029)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KABS

Absolute value key:

0

Use algebraic values of load case *LCNO* in operations.

1

Use absolute values of load case *LCNO* in operations.

Command Default

Use algebraic values.

Notes

Causes absolute values to be used in the load case operations (**LCASE (p. 1019)** or **LCOPER (p. 1026)**). Absolute values are taken prior to assigning a load case factor (**LCFACT (p. 1022)**) and are applied only to defined load cases (**LCDEF (p. 1021)**).

When **LCABS (p. 1018)** operates on nodal-averaged results, it may yield different numerical values compared to the same data stored as element results. For more information, see Nodal-Averaged Results in the *Element Reference*.

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Absolut Value

---

LCASE, *LCNO*

Reads a load case into the database.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCNO

Load case pointer number (**LCDEF (p. 1021)**,STAT). Defaults to 1.

Command Default

Load case 1.

Notes

Reads a load case into the database. Load cases are created as described on the **LCDEF (p. 1021)** or **LCWRITE (p. 1031)** commands. The results portion of the database and the applied forces and displace-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ments are cleared before reading the data in. Absolute values (**LCABS (p. 1018)**) and scale factors (**LCFACT (p. 1022)**) can be applied during the read operation.

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Load Case>Read Load Case

LCCALC

Specifies "Load case settings" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

This command is also valid for rezoning.

Menu Paths

Utility Menu>List>Status>General Postproc>Load Case Calcs

LCCAT, *NL1*, *NL2*

Concatenates multiple lines into one line for mapped meshing.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2

Lines to be concatenated. If *NL1* = ALL, *NL2* is ignored and all selected lines (**LSEL (p. 1074)**) are concatenated. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* is ignored).

Notes

Concatenates multiple, adjacent lines (the input lines) into one line (the output line) in preparation for mapped meshing. An area that contains too many lines for mapped meshing can still be mapped

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

meshed if some of the lines in that area are first concatenated (see Meshing Your Solid Model in the *Modeling and Meshing Guide* for details on mapped meshing restrictions).

LCCAT (p. 1020) is meant to be used solely for meshing and cannot be used for any other purposes. Specifically, (a) the output line and any areas that have the output line on their line list (**ALIST (p. 113)**) cannot be used as input to any other solid modeling operation (not even another **LCCAT (p. 1020)** command); and (b) the output line cannot accept solid model boundary conditions (**DL (p. 545)**, **SFL (p. 1774)**).

The output line will take on the element divisions of the input lines and will not accept element divisions that are directly assigned (**LESIZE (p. 1041)**). The output line from the **LCCAT (p. 1020)** operation will be coincident with the input lines and the input lines will be retained. Consider the **LCOMB (p. 1025)** command instead of **LCCAT (p. 1020)** if you wish to delete the input lines and if the lines to be combined have similar slopes at the common keypoint(s). When an **LCCAT (p. 1020)** command is issued, area line lists (**ALIST (p. 113)**) that contain *all* of the input lines will be updated so that the area line lists refer to the output line instead of the input lines. Deletion of the output line (**LDELE (p. 1033)**) effectively reverses the **LCCAT (p. 1020)** operation and restores area line lists to their original condition.

You can use the **LSEL (p. 1074)** command to select lines that were created by concatenation, and then follow it with an **LDELE (p. 1033)**,ALL command to delete them. Also see Meshing Your Solid Model in the *Modeling and Meshing Guide* for a discussion on how to easily select and delete concatenated lines in one step.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>Concatenate>Lines

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Concatenate>Lines

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Geometry Controls>Concatenate Lines

---

LCDEF, *LCNO*, *LSTEP*, *SBSTEP*, *KIMG*

Creates a load case from a set of results on a results file.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCNO

Arbitrary pointer number (1-99) to be assigned to the load case specified by *LSTEP*, *SBSTEP* and by the **FILE (p. 752)** command. Defaults to 1 + previous value.

LSTEP

Load step number to be defined as the load case. Defaults to one.

SBSTEP

Substep number. Defaults to the last substep of the load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KIMG

Used only with results from complex analyses:

0

Use real part of complex solution

1

Use imaginary part.

Notes

Creates a load case by establishing a pointer to a set of results on a results file (written during the analysis solution phase). This pointer (*LCNO*) can then be used on the **LCASE (p. 1019)** or **LCOPER (p. 1026)** commands to read the load case data into the database.

Issue **LCDEF (p. 1021)**,ERASE to delete *all* load case pointers (and *all* load case files, if any). Issue **LCDEF (p. 1021)**,*LCNO*,ERASE to delete only the specific load case pointer *LCNO* (and its file, if any). With the ERASE options, all pointers are deleted; however only files with the default extension (**LCWRITE (p. 1031)**) are deleted. Issue **LCDEF (p. 1021)**,STAT for status of all selected load cases (**LCSEL (p. 1029)**), or **LCDEF (p. 1021)**,STAT,ALL for status of all load cases. The **STAT (p. 1861)** command may be used to list all load cases. See also **LCFILE (p. 1023)** to establish a pointer to a set of results on a load case file (written by **LCWRITE (p. 1031)**). Harmonic element data read from a result file load case is stored at the zero-degree position.

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Load Case>Create Load Case

Main Menu>General Postproc>Load Case>Erase Load Case

Main Menu>General Postproc>Load Case>List Load Cases

LCFACT, *LCNO*, *FACT*

Defines scale factors for load case operations.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCNO

Load case pointer number. If ALL, apply to all selected load cases (**LCSEL (p. 1029)**).

FACT

Scale factor applied to load case *LCNO*. Blank defaults to 1.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

All factors are 1.0.

Notes

Defines scale factors to be used in the load case operations (**LCASE (p. 1019)** or **LCOPER (p. 1026)**). Scale factors are applied after an absolute value operation (**LCABS (p. 1018)**) and are applied only to defined load cases (**LCDEF (p. 1021)**).

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Scale Factor

---

LCFILE, *LCNO*, *Fname*, *Ext*, --

Creates a load case from an existing load case file.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCNO

Arbitrary (1-99) pointer number assigned to this load case.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to the *LCNO* value preceded by an" L" (for values 10–99) or by an" L0" (for values 1–9).

--

Unused field.

Notes

Creates a load case by establishing a pointer to an existing load case file (**LCWRITE (p. 1031)**). This pointer (*LCNO*) can then be used on the **LCASE (p. 1019)** or **LCOPER (p. 1026)** commands to read the load

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

case data into the database. This command is typically used to reestablish load case pointers in a new Mechanical APDL session (pointers are not saved on the database file), or when more than one pointer to a single load case is desired. See the **LCDEF (p. 1021)** command for status and erase operations. See also **LCDEF (p. 1021)** to establish a pointer to a set of results on a results file (written during the analysis solution phase).

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Load Case>Create Load Case

LCLEAR, *NL1*, *NL2*, *NINC*

Deletes nodes and line elements associated with selected lines.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2*, *NINC

Delete mesh for lines *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and the mesh for all selected lines (**LSEL (p. 1074)**) is deleted. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

Notes

Deletes *all* nodes and line elements associated with selected lines (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed lines and nodes associated with non-line elements will not be deleted. Attributes assigned as a result of **LATT (p. 1014)** are maintained. In the program's response to the command, if a line or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LCOMB, *NL1*, *NL2*, *KEEP*

Combines adjacent lines into one line.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the first line to be combined. If *NL1* = ALL, *NL2* is ignored and all selected lines (**LSEL (p. 1074)**) are combined. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* is ignored).

NL2

Number of the second line to be combined.

KEEP

Specifies whether to keep the input entities:

0

Delete lines *NL1* and *NL2* and their common keypoint. Keypoints will not be deleted if they are meshed or if they are attached to other lines. Lines will not be deleted if they are attached to different areas.

1

Keep *NL1*, *NL2*, and their common keypoint. (The common keypoint will *not* be attached to the output line.)

Notes

Combines adjacent lines into one line (the output line). This operation will effectively "undo" the **LDIV (p. 1034)** operation. Line divisions are set to zero (use **LESIZE (p. 1041)**, etc. to modify). Lines attached to the same area(s) can also be combined. See also the **LCCAT (p. 1020)** command for line concatenation capability.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LCOPER, *Oper*, *LCASE1*, *Oper2*, *LCASE2*, *SweepANG*

Performs load case operations.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Valid operations are:

ZERO

Zero results portion of database (*LCASE1* ignored).

SQUA

Square database values (*LCASE1* ignored).

SQRT

Square root of database (absolute) values (*LCASE1* ignored).

LPRIN

Recalculate line element principal stresses (*LCASE1* ignored). Stresses are as shown for the NMISC items of the **ETABLE (p. 687)** command for the specific line element type.

ADD

Add *LCASE1* to database values.

SUB

Subtract *LCASE1* from database values.

SRSS

Square root of the sum of the squares of database and *LCASE1*.

MIN

Compare and save in database the algebraic minimum of database and *LCASE1*.

MAX

Compare and save in database the algebraic maximum of database and *LCASE1*.

ABMN

Compare and save in database the absolute minimum of database and *LCASE1* (based on magnitudes, then apply the corresponding sign).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ABMX

Compare and save in database the absolute maximum of database and *LCASE1* (based on magnitudes, then apply the corresponding sign).

LCASE1

First load case in the operation (if any). See *LCNO* of the **LCDEF (p. 1021)** command. If ALL, repeat operations using all selected load cases (**LCSEL (p. 1029)**).

Oper2

Valid operations are:

MULT

Multiplication: *LCASE1***LCASE2*

CPXMAX

This option does a phase angle sweep to calculate the maximum of derived stresses, equivalent strain, and principal strains for a complex solution where *LCASE1* is the real part and *LCASE2* is the imaginary part. The *Oper* field is not applicable with this option. Also, the **LCABS (p. 1018)** and **SUMTYPE (p. 1880)** commands have no effect on this option. The value of S3 will be a minimum. Absolute maximum is obtained for component quantity. This option does not apply to derived displacement amplitude (USUM). Load case writing (**LCWRITE (p. 1031)**) is not supported. See POST1 and POST26 – Complex Results Postprocessing in the *Theory Reference* for more information.

LCASE2

Second load case. Used only with *Oper2* operations.

SweepANG

Sweep angle increment in degrees for phase sweep. Used only with *Oper2* = CPXMAX. (Default = 1°)

Notes

LCOPER (p. 1026) operates on the database and one or two load cases according to:

Database = Database *Oper* (*LCASE1 Oper2 LCASE2*)

where operations *Oper* and *Oper2* are as described above.

Absolute values and scale factors may be applied to the load cases before the operations (**LCABS (p. 1018)**, **LCFACT (p. 1022)**). If *LCASE1* is not specified, only operation *Oper* is performed on the current database. If *LCASE2* is specified, operation *Oper2* is performed *before* operation *Oper*. If *LCASE2* is not specified, operation *Oper2* is ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution items not contained (**OUTRES (p. 1336)**) in either the database or the applicable load cases will result in a null item during a load case operation. Harmonic element data read from a result file load case are processed at zero degrees.

Load case combinations must be performed on load cases from the same results file. Combining load cases from different results files is not supported.

Load case combinations of element-based solutions are performed in the solution coordinate system, and the data resulting from load case combinations are stored in the solution coordinate system. The resultant data are then transformed to the active results coordinate system (**RSYS (p. 1639)**) when listed or displayed. Except in the cases of *Oper* = LPRIN, ADD, or SUB, you must use **RSYS (p. 1639)**,SOLU to list or display results. In the case of layered elements, the layer (**LAYER (p. 1015)**) must also be specified.

If nodal-averaged results are a part of the load case combination, the resulting nodal-averaged result data are in the global Cartesian coordinate system. Furthermore, the resulting numerical values may differ from the same **LCOPER (p. 1026)** operation performed on the same data stored as element results. The SQUA, SQRT, LPRIN, and SRSS operations are not valid when applied to nodal averaged results. For more information, see Nodal-Averaged Results in the *Element Reference*.

Use the **FORCE (p. 772)** command prior to any combination operation to correctly combine the requested force type.

If Oper2=CPXMAX, the derived stresses and strain calculation do not apply to line elements.

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

For cyclic symmetry (**CYCLIC (p. 462)**), **LCOPER (p. 1026)** operates on the raw base and duplicate sector values. It cannot be used on the expanded values (**/CYCEXPAND (p. 454)**).

Menu Paths

Main Menu>General Postproc>Load Case>Add

Main Menu>General Postproc>Load Case>Line Elem Stress

Main Menu>General Postproc>Load Case>Min & Max

Main Menu>General Postproc>Load Case>Square

Main Menu>General Postproc>Load Case>Square Root

Main Menu>General Postproc>Load Case>SRSS

Main Menu>General Postproc>Load Case>Subtract

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LCSEL, *Type*, *LCMIN*, *LCMAX*, *LCINC*

Selects a subset of load cases.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set.

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

LCMIN

Minimum value of load case pointer range.

LCMAX

Maximum value of load case pointer range. *LCMAX* defaults to *LCMIN*.

LCINC

Value increment within range. Defaults to 1. *LCINC* cannot be negative.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

All load cases are selected.

Notes

Selects a subset of load cases for other operations. For example, to select a new set of load cases based on load cases 1 through 7, use **LCSEL (p. 1029)**,S,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **LCFACT (p. 1022)**, **LCABS (p. 1018)**, **LCOPER (p. 1026)**, etc. Load cases are flagged as selected and unselected; no load case pointers (**LCDEF (p. 1021)**, **LCWRITE (p. 1031)**, **LCFILE (p. 1023)**) are actually deleted from the database.

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Sele Ld Cases

---

LCSL, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Divides intersecting lines at their point(s) of intersection.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NL3, . . . , NL9

Numbers of lines to be intersected. If *NL1* = ALL, *NL2* to *NL9* are ignored and the intersection of all selected lines is found. If *NL1* = P, use graphical picking to specify lines (*NL2* to *NL9* are ignored).

Notes

Divides intersecting (classifies) lines at their point(s) of intersection. The original lines (and their corresponding keypoint(s)) will be deleted by default. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LCSUM, *Lab*

Specifies whether to process non-summable items in load case operations.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Combination option

(blank)

Only combine summable items [default].

ALL

Combine all items including non summable items.

Notes

Allows non-summable items (e.g. plastic strains) to be included in load combinations. Issue **LCSUM (p. 1031)**,ALL before the first load case operation (**LC*XX*** command). May also be used to include nonsummable items in the appending of a results file (**RAPPND (p. 1570)** command).

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

LCWRITE, *LCNO*, *Fname*, *Ext*, --

Creates a load case by writing results to a load case file.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCNO

Arbitrary pointer number (1-99) to be assigned to this load case.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ext

Filename extension (eight-character maximum).

The extension defaults to the *LCNO* value preceded by an "L" (for values 10–99) or by an "L0" (for values 1–9).

--

Unused field.

Notes

Creates a load case by writing the results data in the database to a load case file. The database remains unchanged by this operation. A pointer is also established to the written set of results on the load case file. This pointer (*LCNO*) can then be used on the **LCASE (p. 1019)** or **LCOPER (p. 1026)** commands to read the load case data into the database. By default, only summable results data (such as displacements, stresses, elastic strains) and constant results data (such as volume) are written to the load case file unless requested (**LCSUM (p. 1031)** command). Non-summable results data (such as plastic strains, strain energy), boundary conditions, and nodal loads are not written to the load case file. The load case file may be named by default or by a user name. Rewriting to the same file overwrites the previous data. See the **LCDEF (p. 1021)** command for status and erase operations.

For details on using load case combination, see Creating and Combining Load Cases in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Load Case>Write Load Case

LCZERO

Zeroes the results portion of the database.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Often used before the **LCOPER (p. 1026)** command. Same as **LCOPER (p. 1026)**,ZERO.

Menu Paths

Main Menu>General Postproc>Load Case>Zero Load Case

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LDELE, *NL1*, *NL2*, *NINC*, *KSWP*

Deletes unmeshed lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2*, *NINC

Delete lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines (**LSEL (p. 1074)**) are deleted. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

KSWP

Specifies whether keypoints are also to be deleted:

0

Delete lines only.

1

Delete lines, as well as keypoints attached to lines but not attached to other lines.

Notes

A line attached to an area cannot be deleted unless the area is first deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Concatenate>Del Concats>Lines

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>Del Concats>Lines

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>Del Concats>Lines

Main Menu>Preprocessor>Modeling>Delete>Del Concats>Lines

Main Menu>Preprocessor>Modeling>Delete>Line and Below

Main Menu>Preprocessor>Modeling>Delete>Lines Only

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LDIV,*NL1*,*RATIO*,*PDIV*,*NDIV*,*KEEP*

Divides a single line into two or more lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the line to be divided. If negative, assume *P1* (see below) is the second keypoint of the line instead of the first for *RATIO*. If ALL, divide all selected lines (**LSEL (p. 1074)**). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

RATIO

Ratio of line length *P1-PDIV* to line length *P1-P2*. Must be between 0.0 and 1.0. Input ignored if *NDIV* > 2.

PDIV

Number to be assigned to keypoint generated at division location (defaults to lowest available keypoint number (**NUMSTR (p. 1301)**)). Input ignored if *NL1* = ALL or *NDIV* > 2. If *PDIV* already exists and lies on line *NL1*, divide line at *PDIV* (*RATIO* must also be 0.0). If *PDIV* already exists and does not lie on line *NL1*, *PDIV* is projected and moved to the nearest point on line *NL1* (if possible). *PDIV* cannot be attached to another line, area, or volume.

NDIV

The number of new lines to be generated from old line (defaults to 2).

KEEP

Specifies whether to keep the input entities:

0

Modify old line to use new keypoints and slopes.

1

Do not modify old line. New lines will overlay old line and have unique keypoints.

Notes

Divides a single line *NL1* (defined from keypoint *P1* to keypoint *P2*) into two or more lines. Line *NL1* becomes the new line beginning with keypoint *P1* and new lines are generated ending at keypoint *P2*. If the line is attached to an area, the area will also be updated. Line divisions are set to zero (use **LESIZE (p. 1041)**, etc. to modify).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line into 2 Ln's

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line into N Ln's

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Lines w/ Options

---

LDRAG, *NK1, NK2, NK3, NK4, NK5, NK6, NL1, NL2, NL3, NL4, NL5, NL6*

Generates lines by sweeping a keypoint pattern along path.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NK1, NK2, NK3, . . . , NK6

List of keypoints in the pattern to be dragged (6 maximum if using keyboard entry). If *NK1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NK1* = ALL, all selected keypoints (except those that define the drag path) will be swept along the path. A component name may also be substituted for *NK1*.

NL1, NL2, NL3, . . . , NL6

List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines.

Notes

Generates lines (and their corresponding keypoints) by sweeping a given keypoint pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NL1*, *NL2*, etc.). If the drag path is a single line (*NL1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves. Keypoint and line numbers are automatically assigned (beginning with the lowest available values (**NUMSTR (p. 1301)**)). For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Keypoints>Along Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LDREAD, *Lab*, *LSTEP*, *SBSTEP*, *TIME*, *KIMG*, *Fname*, *Ext*, --

Reads results from the results file and applies them as loads.

SOLUTION (p. 38): FE Constraints (p. 47)
SOLUTION (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid load label:

TEMP

Temperatures from a thermal analysis are applied as body force nodal loads (**BF (p. 221)**) in a structural analysis or other type of analysis.

When used in conjunction with *KIMG* = 1 or *KIMG* = 2, temperatures can be applied to a subsequent thermal analysis as nodal loads (**D (p. 483)**) or initial conditions (**IC (p. 921)**), respectively.

See the "Notes" (p. 1039) section for details on transferring temperatures from layered thermal shell elements (SHELL131, SHELL132) and layered thermal solid elements (SOLID278, SOLID279).

FORC

Forces from an electromagnetic analysis are applied as force loads (**F (p. 729)**) in a structural analysis. **LDREAD (p. 1036)**,FORC reads coupling forces. See the discussion on force computation in the *Low-Frequency Electromagnetic Analysis Guide*.

For a full harmonic magnetic analysis, FORC represents the time-averaged force (use in conjunction with *KIMG* = 2). Values are in the nodal coordinate system for the force loads (**F (p. 729)**).

HGEN

Heat generations from an electromagnetic analysis are applied as body-force loads (**BFE (p. 237)**) in a thermal analysis. For a full harmonic analysis, HGEN represents the time-averaged heat generation load (use in conjunction with *KIMG* = 2).

JS

Source current density from a current-conduction analysis are applied as body-force loads (**BFE (p. 237)**). Values are in the global Cartesian coordinate system.

EF

Electric field element centroid values from an electrostatic analysis are applied as body-force loads (**BFE (p. 237)**) in a magnetic analysis. Values are in the global Cartesian coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REAC

Reaction loads from any analysis are applied as force loads (**F (p. 729)**) in any analysis. Values are in the nodal coordinate system.

CONC

Concentrations from a diffusion analysis are applied to a subsequent diffusion analysis as nodal loads (**D (p. 483)**) or initial conditions (**IC (p. 921)**) when used in conjunction with *KIMG*=1 or *KIMG*=2, respectively.

VMEN

Mean flow velocities from a static mean flow analysis are applied to a subsequent harmonic or modal solution of the convective wave equation as body-force loads (**BF (p. 221)**).

VOLT

Voltages from an electric, electrostatic, or electromagnetic analysis are applied to a subsequent electric, electrostatic, or electromagnetic analysis as nodal loads (**D (p. 483)**) when *KIMG* = 1 or as initial conditions (**IC (p. 921)**) when *KIMG* = 2.

LSTEP

Load step number of the data set to be read. Defaults to 1. If LAST, ignore *SBSTEP* and *TIME* and read the last data set.

SBSTEP

Substep number (within *LSTEP*). If zero (or blank), *LSTEP* represents the last substep of the load step.

TIME

Time-point identifying the data set to be read. Used only if both *LSTEP* and *SBSTEP* are zero (or blank). If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. If *TIME* is beyond the last time point on the file, use the last time point.

KIMG

When used with results from harmonic analyses (**ANTYPE (p. 162)**,HARMIC) *KIMG* establishes which set of data to read:

0

Read the real part of the solution. Valid also for *Lab* = EHFLU to read in time-average heat flux.

1

Read the imaginary part of the solution.

2

Calculate and read the time-average part. Meaningful for *Lab* = HGEN or FORC.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When used with the PRES label, *KIMG* represents the shell element face on which to apply the pressure:

1

Apply pressure to face 1

2

Apply pressure to face 2

When used with the TEMP label, *KIMG* indicates how temperatures are to be applied.

0

Apply temperatures as body loads (**BF (p. 221)**)

1

Apply temperatures as nodal loads (**D (p. 483)**)

2

Apply temperatures as initial conditions (**IC (p. 921)**)

When used with the CONC label, *KIMG* indicates how concentrations are to be applied.

1

Apply concentrations as nodal loads (**D (p. 483)**)

2

Apply concentrations as initial conditions (**IC (p. 921)**)

When used with the VOLT label, *KIMG* indicates how voltages are to be applied.

1

Apply voltages as nodal loads (**D (p. 483)**)

2

Apply voltages as initial conditions (**IC (p. 921)**)

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The extension defaults to RST (or RMF for a static mean flow analysis) if *Fname* is blank.

- -

Unused field.

Notes

The **LDREAD (p. 1036)** command reads results data from the results file and applies them as loads.

The command can also apply results from an analysis defined with one physics environment as loads on a second analysis using a different physics environment. Results values are applied as loads for field-coupling effects (for example, output temperatures from a thermal analysis as input to a structural analysis).

The command works based on the assumption that the meshes have not changed.

Nodal loads are applied only to selected nodes. Element loads are applied only to selected elements. Element surface loads are applied only to selected elements where all face nodes for that surface are selected.

To assure proper distribution of the surface loads, select *only* the nodes on the element face where the surface load is to be applied.

Scaling and accumulation specifications are applied as the loads are read via the following commands:

- **BFCUM (p. 233)** for body-force loads. (Heat-generation loads are not accumulated.)
- **SFCUM (p. 1758)** for surface loads.
- **FCUM (p. 739)** for force loads.

List the results via the appropriate list command:

- **BFLIST (p. 255)** or **BFELIST (p. 244)** for body-force loads.
- **SFELIST (p. 1770)** for surface loads.
- **FLIST (p. 766)** for force loads.

Values may be redefined after being read by issuing **LDREAD (p. 1036)** again with a different load step and substep, or time value.

This command is also valid in PREP7.

Transferring Temperature Output from SHELL131 and SHELL132

If a thermal analysis uses SHELL131 or SHELL132 thermal shell elements, temperatures can be transferred as body force element loads (**BFE (p. 237)**). In most cases, only the top and bottom temperatures from SHELL131 and SHELL132 are used by the structural shell elements; any interior temperatures are ignored. However, *all* temperatures are used by SHELL181 having section input, and SHELL281 having section input; for these elements, therefore, the number of temperature points at a node generated in the thermal model must match the number of temperature points at a node needed by the structural model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using SHELL131 or SHELL132 information for the **LDREAD (p. 1036)** operation, all element types should specify the same set of thermal degrees of freedom.

Transferring Temperature Output from SOLID278 and SOLID279

If a thermal analysis uses SOLID278 or SOLID279 thermal solid elements, the temperatures are available either at the nodes (KEYOPT(3) = 0) or at the nodes and layers (KEYOPT(3) = 1 or 2). Under normal circumstances, only the nodal temperatures are transferred to the structural elements.

However, if the structural elements are layered solids (KEYOPT(3) = 1 for SOLSH190, SOLID185, SOLID186) and the thermal elements have KEYOPT(3) = 1 or 2 (layered solid) and KEYOPT(8) = 1 (store data for all layers), then the layer temperatures are transferred to the structural elements. If the number of layers do not match, the algorithm reverts back to nodal temperature transfer.

KIMG = 0 (body loads) is the only valid mode for layered temperature transfer.

Examples

Thermal-Stress Example: Load Transfer Coupled-Field Analysis with One-way Coupling

Induction Heating Example: Load Transfer Coupled-Field Analysis with Two-way Coupling

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>From Therm Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Heat Generat>From Mag Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Initial Condit'n>Temp from ANSYS

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>From Therm Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>From Elec An

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>Electric Field>From Elec An

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>From Mag Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Force/Moment>From Reactions

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>From Fluid Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Temperature>From Therm Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>From Fluid Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>From EMAG Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>From Fluid Analy

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Generat>From Mag Analy

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>From ANSYS

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>From Therm Analy

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Heat Generat>From Mag Analy

Main Menu>Solution>Define Loads>Apply>Initial Condit'n>Temp from ANSYS

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>From Therm Analy

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>AppCurrDens>From Elec An

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>Electric Field>From Elec An

Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>From Mag Analy

Main Menu>Solution>Define Loads>Apply>Structural>Force/Moment>From Reactions

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>From Fluid Analy

Main Menu>Solution>Define Loads>Apply>Structural>Temperature>From Therm Analy

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>From Fluid Analy

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>From EMAG Analy

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>From Fluid Analy

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Generat>From Mag Analy

Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>From ANSYS

LESIZE,*NL1*,*SIZE*,*ANGSIZ*,*NDIV*,*SPACE*,*KFORC*,*LAYER1*,*LAYER2*,*KYNDIV*

Specifies the divisions and spacing ratio on unmeshed lines.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the line to be modified. If ALL, modify all selected lines (**LSEL (p. 1074)**). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

SIZE

If *NDIV* is blank, *SIZE* is the division (element edge) length. The number of divisions is automatically calculated from the line length (rounded upward to next integer). If *SIZE* is zero (or blank), use *ANGSIZ* or *NDIV*.

ANGSIZ

The division arc (in degrees) spanned by the element edge (except for straight lines, which always result in one division). The number of divisions is automatically calculated from the line length (rounded upward to next integer).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NDIV

If positive, *NDIV* is the number of element divisions per line. If -1 (and *KFORC* = 1), *NDIV* is assumed to be zero element divisions per line. TARGE169 with a rigid specification ignores *NDIV* and will always mesh with one element division.

SPACE

Spacing ratio. If positive, nominal ratio of last division size to first division size (if > 1.0, sizes increase, if < 1.0, sizes decrease). If negative, |*SPACE*| is nominal ratio of center division(s) size to end divisions size. Ratio defaults to 1.0 (uniform spacing). For layer-meshing, a value of 1.0 normally is used. If *SPACE* = FREE, ratio is determined by other considerations

KFORC

KFORC 0-3 are used only with *NL1* = ALL. Specifies which selected lines are to be modified.

0

Modify only selected lines having undefined (zero) divisions.

1

Modify all selected lines.

2

Modify only selected lines having fewer divisions (including zero) than specified with this command.

3

Modify only selected lines having more divisions than specified with this command.

4

Modify only nonzero settings for *SIZE*, *ANGSIZ*, *NDIV*, *SPACE*, *LAYER1*, and *LAYER2*. If *KFORC* = 4, blank or 0 settings remain unchanged.

LAYER1

Layer-meshing control parameter. Distance which defines the thickness of the inner mesh layer, *LAYER1*. Elements in this layer are uniformly-sized with edge lengths equal to the specified element size for the line (either through *SIZE* or line-length/*NDIV*). A positive value for *LAYER1* is interpreted as an absolute length, while a negative value in interpreted as a multiplier on the specified element size for the line. As a general rule, the resulting thickness of the inner mesh layer should be greater than or equal to the specified element size for the line. If *LAYER1* = OFF, layer-meshing control settings are cleared for the selected lines. The default value is 0.0

LAYER2

Layer-meshing control parameter. Distance which defines the thickness of the outer mesh layer, *LAYER2*. Elements in this layer transition in size from those in *LAYER1* to the global element size. A positive value of *LAYER2* is interpreted as an absolute length, while a negative value is interpreted

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as a mesh transition factor. A value of *LAYER2* = -2 would indicate that elements should approximately double in size as the mesh progresses normal to *LAYER1*. The default value is 0.0.

KYNDIV

0, No, and Off means that SmartSizing cannot override specified divisions and spacing ratios. Mapped mesh fails if divisions do not match. This defines the specification as "hard".

1, Yes, and On means that SmartSizing can override specified divisions and spacing ratios for curvature or proximity. Mapped meshing can override divisions to obtain required matching divisions. This defines the specification as" soft".

Notes

Defines the number of divisions and the spacing ratio on selected lines. Lines with previously specified divisions may also be changed.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Clr Layers

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Layers>Picked Lines

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>All Lines

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Clr Size

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Copy Divs

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Flip Bias

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Lines>Picked Lines

---

LEXTND,*NL1*,*NK1*,*DIST*,*KEEP*

Extends a line at one end by using its slope.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the line to be extended. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NK1

Number of keypoint at the end of line *NL1* to be extended.

DIST

Distance that the line will be extended.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEEP

Specifies whether to keep the input entities:

0

Modify old line to use new keypoints and slopes.

1

Do not modify old line. New line will overlay old line and have unique keypoints.

Notes

Extends a line at one end by using its slope. Lines may be redefined only if not yet attached to an area. Line divisions are set to zero (use **LESIZE (p. 1041)**, etc. to modify). Note that solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extend Line

LFILLT, *NL1*, *NL2*, *RAD*, *PCENT*

Generates a fillet line between two intersecting lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the first intersecting line. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NL2

Number of the second intersecting line.

RAD

Radius of fillet to be generated. Radius should be less than the lengths of the two lines specified with *NL1* and *NL2*.

PCENT

Number to be assigned to generated keypoint at fillet arc center. If zero (or blank), no keypoint is generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates a fillet line between two intersecting lines *NL1* (*P1-PINT*) and *NL2* (*P2-PINT*). Three keypoints may be generated, two at the fillet tangent points (*PTAN1* and *PTAN2*) and one (optional) at the fillet arc center (*PCENT*). Line *P1-PINT* becomes *P1-PTAN1*, *P2-PINT* becomes *P2-PTAN2*, and new arc line *PTAN1-PTAN2* is generated. Generated keypoint and line numbers are automatically assigned (beginning with the lowest available values (**NUMSTR (p. 1301)**)). Line divisions are set to zero (use **LESIZE (p. 1041)**, etc. to modify).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Line Fillet

LFSURF, *SLINE*, *TLINE*

Generates surface elements overlaid on the edge of existing solid elements and assigns the extra node as the closest fluid element node.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SLINE

Component name for the surface lines of the meshed solid areas. The component name must be 32 characters or less.

TLINE

Component name for the target lines meshed with fluid elements. The component name must be 32 characters or less.

Notes

This command macro is used to generate surface effect elements overlaid on the surface of existing plane elements and, based on proximity, to determine and assign the extra node for each surface element. The underlying areas of the solid region and the fluid lines must be meshed prior to calling this command macro. The active element type must be SURF151 with appropriate settings for KEYOPT(4), KEYOPT(5), KEYOPT(6), and KEYOPT(8).

The surface lines of the solid and the target lines of the fluid are grouped into components and named using the **CM (p. 356)** command. The names must be enclosed in single quotes (for example, '*SLINE*') when the **LFSURF (p. 1045)** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the SURF151 and FLUID116 element types.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid

---

LGEN, *ITIME*, *NL1*, *NL2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional lines from a pattern of lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be > 1 for generation to occur.

NL1*, *NL2*, *NINC

Generate lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines (**LSEL (p. 1074)**). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

DX*, *DY*, *DZ

Keypoint location increments in the active coordinate system (--, Dθ, DZ for cylindrical, --, Dθ, -- for spherical).

KINC

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original lines, if they exist.

1

Do not generate nodes and elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IMOVE

Specifies whether to redefine the existing lines:

0

Generate additional lines as requested with the *ITIME* argument.

1

Move original lines to new position retaining the same keypoint numbers (*ITIME*, *KINC*, and *NOELM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates additional lines (and their corresponding keypoints and mesh) from a given line pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial, etc. Generations which produce lines of a size or shape different from the pattern (that is, radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Note that solid modeling in a toroidal coordinate system is not recommended. New line numbers are automatically assigned (beginning with the lowest available values (**NUMSTR (p. 1301)**)).

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Lines

Main Menu>Preprocessor>Modeling>Move / Modify>Lines

LGLUE, *NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9*

Generates new lines by "gluing" lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NL3, . . . , NL9

Numbers of the lines to be glued. If *NL1* = ALL, all selected lines will be glued (*NL2* to *NL9* will be ignored). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

Notes

Use of the **LGLUE (p. 1047)** command generates new lines by "gluing" input lines. The glue operation redefines the input lines so that they share keypoints at their common ends. The new lines encompass the same geometry as the original lines. This operation is only valid if the intersections of the input lines are keypoints at the ends of those lines. See the *Modeling and Meshing Guide* for an illustration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

The **LGLUE (p. 1047)** command results in the merging of keypoints at the common end of the lines. The keypoints of the lower numbered line will be kept. This means one must be aware of line numbering when multiple **LGLUE (p. 1047)** commands are applied to avoid any "ungluing" of geometry.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Lines

LGWRITE,*Fname*,*Ext*,--,*Kedit*

Writes the database command log to a file.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to LGW if *Fname* and *Ext* are blank.

--

Unused field.

Kedit

Flag to suppress nonessential commands:

NONE

Do not suppress any commands (default).

COMMENT

Write nonessential commands as comments (starting with !).

REMOVE

Do not write nonessential commands or comments.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Writes the database command log to a named file. The database command log contains all commands that were used to create the current database. These commands are recorded in the database as they are issued, and saved in the database file (`File.db`) whenever the database is saved. The **LGWRITE (p. 1048)** command extracts these commands from the database and writes them to a file. Nonessential commands (for listing, graphics displays, help, etc.) can be excluded from the file by using the *Kedit* field. The file resulting from **LGWRITE (p. 1048)** can be used as command input to the program. This command is most useful if the session log file (`File.log`), which is normally saved during an interactive session, has been lost or corrupted.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Write DB Log File

/LIGHT, *WN*, *NUM*, *INT*, *XV*, *YV*, *ZV*, *REFL*

Specifies the light direction for the display window.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

NUM

Ambient or directional light key:

0

Ambient light (default).

1

Directional light.

INT

Light intensity factor (defaults to 0.3 for ambient, 1.0 for directional). This option is valid only for 3D devices).

XV, YV, ZV

Light direction (valid only for *NUM* = 1). The directional light source is parallel to the line from point *XV*, *YV*, *ZV* to the origin, in the global Cartesian system origin. Defaults to the viewing direction (**/VIEW (p. 2131)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REFL

Light reflectance factor (valid only for *NUM* = 1 and 3D devices).

Command Default

Use ambient light.

Notes

Defines the light direction for the window. Use this command only with 3D graphics devices or 2D devices when Z-buffering is used (**/TYPE (p. 2034)**,,(6 or 7)). The ambient light has no direction, only an intensity. You can position the directional light source by defining a point (in the global Cartesian coordinate system) representing a point along the light directional line. This point, and the global Cartesian coordinate system origin, define the line along which the light is positioned looking toward the origin. You can use any point along the light line; for example, both (1.,1.,1.) and (2.,2.,2.) give the same light effect. For 3D graphics devices only, the directional light source also has intensity and reflectance factors.

By choosing the highest intensity ambient light for 3D graphics devices (via **/LIGHT (p. 1049)**,WN,0,1), you can nullify color shading and other effects of directional lighting.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Light Source

LINA, *NL*, *NA*

Finds the intersection of a line with an area.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL

Number of line to be intersected. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NA

Number of area to be intersected.

Notes

Finds the intersection of a line with an area. New lines will be generated where the lines intersect the areas. If the regions of intersection are only points, new keypoints will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Line with Area

---

LINE

Specifies "Lines" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

/LINE, *X1*, *Y1*, *X2*, *Y2*

Creates annotation lines (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X1

Line X starting location (-1.0 < X < 2.0).

Y1

Line Y starting location (-1.0 < Y < 1.0).

X2

Line X ending location (-1.0 < X < 2.0).

Y2

Line Y ending location (-1.0 < Y < 1.0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command defines annotation lines to be written directly onto the display at a specified location.

The command is generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if annotation is used. It is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

All lines are shown on subsequent displays unless the annotation is turned off or deleted. Issue **/LSPEC (p. 1081)** to set the attributes of the line.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

LINES, *N*

Specifies the length of a printed page.

POST26 (p. 58): Listing (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

N

Number of lines per page (defaults to 20). (Minimum allowed = 11).

Command Default

20 lines per page.

Notes

Specifies the length of a printed page (for use in reports, etc.).

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINL, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Finds the common intersection of lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NL3, . . . , NL9

Numbers of lines to be intersected. If *NL1* = ALL, find the intersection of all selected lines and *NL2* to *NL9* are ignored. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

Notes

Finds the common (not pairwise) intersection of lines. The common intersection is defined as the regions shared (in common) by *all* lines listed on this command. New lines will be generated where the original lines intersect. If the regions of intersection are only points, new keypoints will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Lines

LINP, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Finds the pairwise intersection of lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NL3, . . . , NL9

Numbers of lines to be intersected pairwise. If *NL1* = ALL, find the pairwise intersection of all selected lines and *NL2* to *NL9* are ignored. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NL1*.

Notes

Finds the pairwise intersection of lines. The pairwise intersection is defined as any and all regions shared by at least two lines listed on this command. New lines will be generated where the original lines intersect pairwise. If the regions of pairwise intersection are only points, new keypoints will be generated. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Lines

---

LINV, *NL*, *NV*

Finds the intersection of a line with a volume.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL

Number of line to be intersected. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NV

Number of volume to be intersected.

Notes

Finds the intersection of a line with a volume. New lines will be generated where the lines intersect the volumes. If the regions of intersection are only points, new keypoints will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Line with Volume

---

LIST, *LEVEL*

Lists out the sets in the results file.

AUX3 (p. 61): Results Files (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command lists the results set number, the load step, substep, and time step for each set. It also shows all sets marked for deletion.

The **LIST (p. 1054)** command is valid only in the results file editing processor (auxiliary processor AUX3).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

---

*LIST, *Fname*, *Ext*, --

Displays the contents of an external, coded file.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Notes

Displays the contents of an external, coded file. The file to be listed cannot be in use (open) at the time (except for the error file, `Jobname.err`, which may be displayed with ***LIST (p. 1055)**,ERR).

Use caution when you are listing active Mechanical APDL files via the **List> Files> Other** and **File> List> Other** menu paths. File I/O buffer and system configurations can result in incomplete listings unless the files are closed.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LLIST, *NL1*, *NL2*, *NINC*, *Lab*

Lists the defined lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2*, *NINC

List lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL (default), *NL2* and *NINC* are ignored and all selected lines (**LSEL (p. 1074)**) are listed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

Lab

Determines what type of listing is used (one of the following):

(blank)

Prints information about all lines in the specified range.

RADIUS

Prints the radius of certain circular arcs, along with the keypoint numbers of each line. Straight lines, non-circular curves, and circular arcs not internally identified as arcs (which depends upon how each arc is created) will print a radius value of zero.

LAYER

Prints layer-mesh control specifications.

HPT

Prints information about only those lines that contain hard points. HPT is not supported in the GUI.

ORIENT

Prints a list of lines, and identifies any orientation keypoints and any cross section IDs that are associated with the lines. Used for beam meshing with defined orientation nodes and cross sections.

Notes

There are 2 listings for the number of element divisions and the spacing ratio. The first listing shows assignments from **LESIZE (p. 1041)** only, followed by the "hard" key (*KYNDIV*). See **LESIZE (p. 1041)** for more information. The second listing shows *NDIV* and *SPACE* for the existing mesh, if one exists. Whether this existing mesh and the mesh generated by **LESIZE (p. 1041)** match at any given point depends upon meshing options and the sequence of meshing operations.

A "-1" in the "nodes" column indicates that the line has been meshed but that there are no interior nodes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **LATT (p. 1014)** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer (**TYPE (p. 2036)**, **MAT (p. 1111)**, **REAL (p. 1579)**, or **ESYS (p. 685)**) that was active during meshing (and will be reset to zero if the mesh is cleared).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Lines

LMESH, *NL1*, *NL2*, *NINC*

Generates nodes and line elements along lines.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2*, *NINC

Mesh lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines (**LSEL (p. 1074)**) are meshed. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

Notes

Generates nodes and line elements along lines. Missing nodes required for the generated elements are created and assigned the lowest available numbers.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Lines

LNSRCH, *Key*, *LSTOL*, *LStrun*

Activates a line search to be used with Newton-Raphson.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Line search key:

OFF

Do not use a line search.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON

Use a line search. Note, adaptive descent is suppressed when **LNSRCH (p. 1057)** is on unless explicitly requested on the **NROPT (p. 1261)** command. *Having line search on and adaptive descent on at the same time is not recommended.*

AUTO

The program automatically switches line searching ON and OFF between substeps of a load step as needed. This option is recommended.

LSTOL

Line search convergence tolerance (default = 0.5).

LStrun

Truncation key for the line search parameter. Default = OFF, meaning no truncation. To activation truncation, input the number of digits to use after the decimal point for the line search parameter. (See Line Search in the *Theory Reference* for more information.)

Command Default

Line search off, unless contact elements are present.

Notes

Activates a line search to be used with the Newton-Raphson method (**NROPT (p. 1261)**). Line search is an alternative to adaptive descent (see Line Search in the *Theory Reference*).

LNSRCH (p. 1057),AUTO can be very efficient for problems in which **LNSRCH (p. 1057)** is needed at only certain substeps.

You cannot use line search (**LNSRCH (p. 1057)**), automatic time stepping (**AUTOTS (p. 207)**), or the DOF solution predictor (**PRED (p. 1476)**) with the arc-length method (**ARCLEN (p. 175)**, **ARCTRM (p. 177)**). If you activate the arc-length method after you set **LNSRCH (p. 1057)**, **AUTOTS (p. 207)**, or **PRED (p. 1476)**, a warning message appears. If you choose to proceed with the arc-length method, the program disables your line search, automatic time stepping, and DOF predictor settings, and the time step size is controlled by the arc-length method internally.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Line Search

Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Solution>Load Step Opts>Nonlinear>Line Search

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LOCAL, *KCN*, *KCS*, *XC*, *YC*, *ZC*, *THXY*, *THYZ*, *THZX*, *PAR1*, *PAR2*

Defines a local coordinate system by a location and orientation.

DATABASE (p. 13): Coordinate System (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Arbitrary reference number assigned to this coordinate system. Must be greater than 10. A coordinate system previously defined with this number will be redefined.

KCS

Coordinate system type:

0 or CART

Cartesian

1 or CYLIN

Cylindrical (circular or elliptical)

2 or SPHE

Spherical (or spheroidal)

3 or TORO

Toroidal

XC, YC, ZC

Location (in the global Cartesian coordinate system) of the origin of the new coordinate system.

THXY

First rotation about local Z (positive X toward Y).

THYZ

Second rotation about local X (positive Y toward Z).

THZX

Third rotation about local Y (positive Z toward X).

PAR1

Used for elliptical, spheroidal, or toroidal systems. If *KCS* = 1 or 2, *PAR1* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle)). If *KCS* = 3, *PAR1* is the major radius of the torus.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PAR2

Used for spheroidal systems. If *KCS* = 2, *PAR2* = ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle)).

Notes

Defines a local coordinate system by origin location and orientation angles. The local coordinate system is parallel to the global Cartesian system unless rotated. Rotation angles are in degrees and redefine any previous rotation angles. See the **CLOCAL (p. 352)**, **CS (p. 435)**, **CSWPLA (p. 440)**, and **CSKP (p. 438)** commands for alternate definitions. This local system becomes the active coordinate system (**CSYS (p. 441)**). Local coordinate systems may be displayed with the **/PSYMB (p. 1552)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>At Specified Loc

---

LOVLAP, *NL1, NL2, NL3, NL4, NL5, NL6, NL7, NL8, NL9*

Overlaps lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NL3, . . . , NL9

Numbers of lines to be overlapped. If *NL1* = ALL, *NL2* to *NL9* are ignored and all selected lines are overlapped. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1*.

Notes

Overlaps lines. Generates new lines which encompass the geometry of all the input lines. The new lines are defined by the regions of intersection of the input lines, and by the complementary (non-intersecting) regions. See the *Modeling and Meshing Guide* for an illustration. This operation is only valid when the region of intersection is a line. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LPLOT, *NL1*, *NL2*, *NINC*

Displays the selected lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2*, *NINC

Display lines from *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL (default), *NL2* and *NINC* are ignored and display all selected lines (**LSEL (p. 1074)**).

Notes

Mesh divisions on plotted lines are controlled by the *LDIV* option of the **/PSYMB (p. 1552)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Lines

Utility Menu>Plot>Specified Entities>Lines

LPTN, *NL1*, *NL2*, *NL3*, *NL4*, *NL5*, *NL6*, *NL7*, *NL8*, *NL9*

Partitions lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2*, *NL3*, . . . , *NL9

Numbers of lines to be operated on. If *NL1* = ALL, *NL2* to *NL9* are ignored all selected lines are used. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NL1*.

Notes

Partitions lines. Generates new lines which encompass the geometry of all the input lines. The new lines are defined by both the regions of intersection of the input lines and the complementary (non-intersecting) regions. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LREFINE, *NL1*, *NL2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified lines.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1, NL2, NINC

Lines (*NL1* to *NL2* in increments of *NINC*) around which the mesh is to be refined. *NL2* defaults to *NL1*, and *NINC* defaults to 1. If *NL1* = ALL, *NL2* and *NINC* are ignored and all selected lines are used for refinement. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH

Depth of mesh refinement in terms of the number of elements outward from the indicated lines (defaults to 1).

POST

Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF

No postprocessing will be done.

SMOOTH

Smoothing will be done. Node locations may change.

CLEAN

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (Mechanical APDL ignores the *RETAIN* argument when you are refining anything other than a quadrilateral mesh.)

ON

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

The final mesh may include some triangular elements in order to maintain element quality and provide transitioning.

Notes

LREFINE (p. 1062) performs local mesh refinement around the specified lines. By default, the indicated elements are split to create new elements with 1/2 the edge length of the original elements (*LEVEL* = 1).

LREFINE (p. 1062) refines all area elements and tetrahedral volume elements that are adjacent to the specified lines. Any volume elements that are adjacent to the specified lines, but are not tetrahedra (for example, hexahedra, wedges, and pyramids), are not refined.

You cannot use mesh refinement on a solid model that contains initial conditions at nodes (**IC (p. 921)**), coupled nodes (**CP (p. 421)** family of commands), constraint equations (**CE (p. 297)** family of commands), or boundary conditions or loads applied directly to any of its nodes or elements. This applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. For additional restrictions on mesh refinement, see Revising Your Model in the *Modeling and Meshing Guide*.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines

LREVERSE, *LNUM*, *NOEFLIP*

Reverses the normal of a line, regardless of its connectivity or mesh status.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LNUM

Line number of the line whose normal direction is to be reversed. If *LNUM* = ALL, the normals of all selected lines will be reversed. If *LNUM* = P, graphical picking is enabled. A component name may also be substituted for *LNUM*.

NOEFLIP

Indicates whether you want to change the normal direction of the existing elements on the reversed line(s) so that they are consistent with each line's new normal direction.

0

Make the normal direction of existing elements on the reversed line(s) consistent with each line's new normal direction (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Do not change the normal direction of existing elements on the reversed line(s).

Notes

You cannot use the **LREVERSE (p. 1063)** command to change the normal direction of any element that has a body or surface load. We recommend that you apply all of your loads only *after* ensuring that the element normal directions are acceptable.

Real constants (such as nonuniform shell thickness and tapered beam constants) may be invalidated by an element reversal.

For more information, see Revising Your Model in the *Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Reverse Normals>of Lines

---

LROTAT, *NK1, NK2, NK3, NK4, NK5, NK6, PAX1, PAX2, ARC, NSEG*

Generates circular lines by rotating a keypoint pattern about an axis.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NK1, NK2, NK3, . . . , NK6

List of keypoints in the pattern to be rotated (6 maximum if using keyboard entry). If *NK1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NK1* = ALL, all selected keypoints (except *PAX1* and *PAX2*) will define the pattern to be rotated. A component name may also be substituted for *NK1*.

PAX1, PAX2

Keypoints defining the axis about which the keypoint pattern is to be rotated.

ARC

Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360.

NSEG

Number of lines (8 maximum) around circumference. Defaults to minimum required for 90° (maximum) arcs, that is, 4 for 360°, 3 for 270°, etc.

Notes

Generates circular lines (and their corresponding keypoints) by rotating a keypoint pattern about an axis. Keypoint patterns are generated at regular angular locations (based on a maximum spacing of

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

90°). Line patterns are generated at the keypoint patterns. Keypoint and line numbers are automatically assigned (beginning with the lowest available values (**NUMSTR (p. 1301)**)).

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Keypoints>About Axis

---

LSBA, *NL*, *NA*, *SEPO*, *KEEPL*, *KEEPA*

Subtracts areas from lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL*.

NA

Area (or areas, if picking is used) to be subtracted. If ALL, use all selected areas. A component name may also be substituted for *NA*.

SEPO

Behavior if the intersection of the lines and the areas is a keypoint or keypoints:

(blank)

The resulting lines will share keypoint(s) where they touch.

SEPO

The resulting lines will have separate, but coincident keypoint(s) where they touch.

KEEPL

Specifies whether *NL* lines are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NL* lines after **LSBA (p. 1065)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NL* lines after **LSBA (p. 1065)** operation (override **BOPTN (p. 269)** command settings).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEEPA

Specifies whether *NA* areas are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete areas after **LSBA (p. 1065)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep areas after **LSBA (p. 1065)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new lines by subtracting the regions common to both *NL* lines and *NA* areas (the intersection) from the *NL* lines. The intersection can be a line(s) or keypoint(s). If the intersection is a keypoint and *SEPO* is blank, the *NL* line is divided at the keypoint and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL* is divided into two unconnected lines with separate keypoints where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Area

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Area

---

***LSBAC**,*EngineName*,*RhsVector*,*SolVector*,*TransKey*

Performs the solve (forward/backward substitution) of a factorized linear system.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

EngineName

Name used to identify this engine. Must have been previously created using ***LSENGINE (p. 1077)** and factorized using ***LSFACTOR (p. 1078)**.

RhsVector

Name of vector containing the right-hand side (load) vectors as input. Must have been previously defined as a ***VEC (p. 2085)** vector or a ***DMAT (p. 551)** matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SolVector

Name of vector that will contain the solution vectors upon completion. Must be predefined as a ***VEC (p. 2085)** vector or ***DMAT (p. 551)** matrix.

TransKey

Transpose key. Set *TransKey* = TRANS to solve the transposed linear system. If blank, transpose will not be used.

Notes

This command performs forward and back substitution to obtain the solution to the linear matrix equation Ax = b (or $A^T x = b$ if *TransKey* = TRANS). The matrix engine must have been previously defined using ***LSENGINE (p. 1077)**, and the matrix factored using ***LSFACTOR (p. 1078)**.

You can use the ***DMAT (p. 551)**,,,COPY (or ***VEC (p. 2085)**,,,COPY) command to copy the load vector to the solution vector in order to predefine it with the appropriate size.

Menu Paths

This command cannot be accessed from a menu.

LSBL, *NL1*, *NL2*, *SEP0*, *KEEP1*, *KEEP2*

Subtracts lines from lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. Lines specified in this argument are not available for use in the *NL2* argument. If P, graphical picking is enabled (valid only in the GUI) and all remaining fields are ignored. A component name may also be substituted for *NL1*.

NL2

Line (or lines, if picking is used) to subtract. If ALL, use all selected lines (except those included in the *NL1* argument). A component name may also be substituted for *NL2*.

SEP0

Behavior if the intersection of the *NL1* lines and the *NL2* lines is a keypoint or keypoints:

(blank)

The resulting lines will share keypoint(s) where they touch.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SEPO

The resulting lines will have separate, but coincident keypoint(s) where they touch.

KEEP1

Specifies whether *NL1* lines are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NL1* lines after **LSBL (p. 1067)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NL1* lines after **LSBL (p. 1067)** operation (override **BOPTN (p. 269)** command settings).

KEEP2

Specifies whether *NL2* lines are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NL2* lines after **LSBL (p. 1067)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NL2* lines after **LSBL (p. 1067)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new lines by subtracting the regions common to both *NL1* and *NL2* lines (the intersection) from the *NL1* lines. The intersection can be a line(s) or point(s). If the intersection is a point and *SEPO* is blank, the *NL1* line is divided at the point and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL1* is divided into two unconnected lines with separate keypoints where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **LSBL (p. 1067)**,ALL,ALL will have no effect since all the lines (in *NL1*) will be unavailable as *NL2* lines.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Line

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Line

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Lines

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Lines

LSBV, *NL*, *NV*, *SEPO*, *KEEPL*, *KEEPV*

Subtracts volumes from lines.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL

Line (or lines, if picking is used) to be subtracted from. If ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL*.

NV

Volume (or volumes, if picking is used) to be subtracted. If ALL, use all selected volumes. A component name may also be substituted for *NV*.

SEPO

Behavior if the intersection of the *NL* lines and the *NV* volumes is a keypoint or keypoints:

(blank)

The resulting lines will share keypoint(s) where they touch.

SEPO

The resulting lines will have separate, but coincident keypoint(s) where they touch.

KEEPL

Specifies whether *NL* lines are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NL* lines after **LSBV (p. 1069)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NL* lines after **LSBV (p. 1069)** operation (override **BOPTN (p. 269)** command settings).

KEEPV

Specifies whether *NV* volumes are to be deleted:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NV* volumes after **LSBV (p. 1069)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NV* volumes after **LSBV (p. 1069)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new lines by subtracting the regions common to both *NL* lines and *NV* volumes (the intersection) from the *NL* lines. The intersection can be a line(s) or point(s). If the intersection is a point and *SEPO* is blank, the *NL1* line is divided at the point and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL1* is divided into two unconnected lines with separate keypoints where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **LSBL (p. 1067)**,ALL,ALL will have no effect since all the lines (in *NL1*) will be unavailable as *NL2* lines.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by Volume

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by Volume

Main Menu>Preprocessor>Modeling>Operate>Divide>Line by Volu

LSBW, *NL*, *SEPO*, *KEEP*

Subtracts the intersection of the working plane from lines (divides lines).

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL

Line (or lines, if picking is used) to be subtracted from. If *NL* = ALL, use all selected lines. If *NL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be input for *NL*.

SEPO

Behavior of the created boundary.

(blank)

The resulting lines will share keypoint(s) where they touch.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SEPO

The resulting lines will have separate, but coincident keypoint(s).

KEEP

Specifies whether *NL* lines are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NL* lines after **LSBW (p. 1070)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NL* lines after **LSBW (p. 1070)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new lines by subtracting the intersection of the working plane from the *NL* lines. The intersection will be a keypoint(s). The working plane must not be in the same plane as the *NL* line(s). If *SEPO* is blank, the *NL* line is divided and the resulting lines will be connected, sharing a common keypoint where they touch. If *SEPO* is set to SEPO, *NL* is divided into two unconnected lines with separate keypoints. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Areas that completely contain the input lines will be updated if the lines are divided by this operation.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Line by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Line by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Divide>Line by WrkPlane

---

LSCLEAR, *Lab*

Clears loads and load step options from the database.

SOLUTION (p. 38): Load Step Operations (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Label identifying the data to be cleared:

SOLID

Delete only solid model loads.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FE

Delete only finite element loads.

INER

Delete only inertia loads (**ACEL (p. 92)**, etc.).

LFACT

Initialize only load factors (on **DCUM (p. 502)**, **FCUM (p. 739)**, **SFCUM (p. 1758)**, etc.).

LSOPT

Initialize only load step options.

ALL

Delete all loads and initialize all load step options and load factors.

Notes

Loads are deleted, and load step options are initialized to their default values.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All F.E. Loads
Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Inertia Lds
Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Loads & Opts
Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All SolidMod Lds
Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Reset Factors
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>All
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Areas
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Elements
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Keypoints
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Lines
Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Nodes
Main Menu>Solution>Define Loads>Delete>All Load Data>All F.E. Loads
Main Menu>Solution>Define Loads>Delete>All Load Data>All Inertia Lds
Main Menu>Solution>Define Loads>Delete>All Load Data>All Loads & Opts
Main Menu>Solution>Define Loads>Delete>All Load Data>All SolidMod Lds
Main Menu>Solution>Define Loads>Settings>Replace vs Add>Reset Factors

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LSDELE, *LSMIN*, *LSMAX*, *LSINC*

Deletes load step files.

SOLUTION (p. 38): Load Step Operations (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSMIN*, *LSMAX*, *LSINC

Range of load step files to be deleted, from *LSMIN* to *LSMAX* in steps of *LSINC*. *LSMAX* defaults to *LSMIN*, and *LSINC* defaults to 1. If *LSMIN* = ALL, all load step files are deleted (and *LSMAX* and *LSINC* are ignored). The load step files are assumed to be named `Jobname.S`*n*, where *n* is a number assigned by the **LSWRITE (p. 1087)** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for numbers > 99.

Notes

Deletes load step files in the current directory (written by the **LSWRITE (p. 1087)** command).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Delete LS Files

Main Menu>Solution>Define Loads>Operate>Delete LS Files

***LSDUMP**, *EngineName*, *FileName*

Dumps a linear solver engine to a binary File.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

EngineName

Name used to identify this engine. Must have been previously created using ***LSENGINE (p. 1077)** and factorized using ***LSFACTOR (p. 1078)**.

FileName

Name of the file to create.

Notes

Dumps a previously factorized linear solver system to a binary file. Only LAPACK and BCS linear solvers can be used with this feature. The Linear Solver can later be restored with the ***LSRESTORE (p. 1083)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A BCS Sparse Solver can be dumped only if uses the *INCORE* memory option (see **BCSOPTION (p. 218)**).

Menu Paths

This command cannot be accessed from a menu.

LSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

Selects a subset of lines.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item

Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to LINE.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

VMIN

Minimum value of item range. Ranges are line numbers, coordinate values, attribute numbers, etc., as appropriate for the item. If *VMIN* = 0.0, a tolerance of ±1.0E-6 is used, or ±0.005 x *VMIN* if *VMIN* = *VMAX*. A component name (as specified on the **CM (p. 356)** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, ESYS, or NDIV and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **LLIST (p. 1056)** command for a discussion of signed attributes.

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN*.

VINC

Value increment within range. Used only with integer ranges (such as for line numbers). Defaults to 1. *VINC* cannot be negative.

KSWP

Specifies whether only lines are to be selected:

0

Select lines only.

1

Select lines, as well as keypoints, nodes, and elements associated with selected lines. Valid only with *Type* = S.

Command Default

All lines are selected.

Notes

Selects lines based on values of a labeled item and component. For example, to select a new set of lines based on line numbers 1 through 7, use **LSEL (p. 1074)**,S,LINE,,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **LLIST (p. 1056)**,ALL. Only data identified by line number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *Item* = LCCA, the command selects only those lines that were created by concatenation. The *KSWP* field is processed, but the *Comp*, *VMIN*, *VMAX*, and *VINC* fields are ignored.

If *Item* = HPT, the command selects only those lines that contain hard points.

Item = RADIUS is only valid for lines that are circular arcs.

LSEL (p. 1074) is valid in any processor.

For selections based on non-integer numbers (coordinates, results, etc.), items that are within the range *VMIN* -*Toler* and *VMAX* +*Toler* are selected. The default tolerance *Toler* is based on the relative values of *VMIN* and *VMAX* as follows:

- If *VMIN* = *VMAX*, *Toler* = 0.005 x *VMIN*.
- If *VMIN* = *VMAX* = 0.0, *Toler* = 1.0E-6.
- If *VMAX* ≠ *VMIN*, *Toler* = 1.0E-8 x (*VMAX* - *VMIN*).

Use the **SELTOL (p. 1710)** (p. 1710) command to override this default and specify *Toler* explicitly.

Table 193: LSEL - Valid Item and Component Labels

Valid Item and Component Labels LSEL (p. 1074), *Type, Item, Comp, VMIN, VMAX, VINC, KSWP*

Item	Comp	Description
LINE		Line number.
EXT		Line numbers on exterior of selected area (ignore remaining fields).
LOC	X,Y,Z	X, Y, or Z center location in the active coordinate system.
TAN1	X,Y,Z	Unit vector component of outward tangent at beginning of line.
TAN2	X,Y,Z	Unit vector component of outward tangent at end of line.
NDIV		Number of divisions within the line.
SPACE		Spacing ratio of line divisions.
MAT		Material number associated with the line.
TYPE		Element type number associated with the line.
REAL		Real constant set number associated with the line.
ESYS		Element coordinate system associated with the line.
SEC		Cross section ID number. (**SECNUM (p. 1685)**)
LENGTH		Length of the line.
RADIUS		Radius of the line.
HPT		Line number (selects only lines with associated hard points).
LCCA		Concatenated lines (selects only lines that were created by concatenation (**LCCAT (p. 1020)**)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>Select>Entities

---

*LSENGINE, *Type*,*EngineName*,*Matrix*,*Option*

Creates a linear solver engine.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Type

Specifies the algorithm to be used:

DSS --

MKL sparse linear solver.

LAPACK --

LAPACK dense matrix linear solver (default if applied to dense matrices).

DSP --

Distributed sparse solver (default for sparse matrices).

EngineName

Name used to identify this engine. Must be specified.

Matrix

Name of the matrix to solve.

Option

Option to control the memory mode of the DSS solver (used only if *Type* = DSS):

INCORE --

In-core memory mode.

OUTOFCORE --

Out-of-core memory mode.

Notes

This command creates a linear solver engine.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The DSS and DSP solvers can only be used with sparse matrices. For dense matrices, use the LAPACK solver.

Menu Paths

This command cannot be accessed from a menu.

---

*LSFACTOR, *EngineName*, *Option*

Performs the numerical factorization of a linear solver system.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

EngineName

Name used to identify this engine. Must have been previously created using ***LSENGINE (p. 1077)**.

Option

Option to invert the matrix, used only with an LAPACK engine (***LSENGINE (p. 1077)**,LAPACK):

INVERT --

Invert the matrix.

Notes

Performs the computationally intensive, memory intensive factorization of a matrix specified by ***LSENGINE (p. 1077)**, using the solver engine also specified by ***LSENGINE (p. 1077)**.

Menu Paths

This command cannot be accessed from a menu.

LSLA, *Type*

Selects those lines contained in the selected areas.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of line select:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

LSLK, *Type*, *LSKEY*

Selects those lines containing the selected keypoints.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of line select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LSKEY

Specifies whether all contained line keypoints must be selected (**KSEL (p. 995)**):

0

Select line if any of its keypoints are in the selected keypoint set.

1

Select line only if all of its keypoints are in the selected keypoint set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

LSOPER

Specifies "Load step operations" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Load Step Operations

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/LSPEC, *LCOLOR*, *LINSTL*, *XLNWID*

Specifies annotation line attributes (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LCOLOR

Line color:

0

Black

1

Red-Magenta

2

Magenta

3

Blue-Magenta

4

Blue

5

Cyan-Blue

6

Cyan

7

Green-Cyan

8

Green

9

Yellow-Green

10

Yellow

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11

Orange

12

Red

13

Dark Gray

14

Light Gray

15

White

LINSTL

Line style:

0

Solid line.

1

Dashed line.

XLNWID

Line width multiplier (1.0 to 20.0). Defaults to 1.0.

Notes

This command specifies annotation line attributes to control certain characteristics of the lines created via the **/LINE (p. 1051)**, **/LARC (p. 1011)**, **/LSYMBOL (p. 1088)**, **/POLYGON (p. 1460)**, **/PMORE (p. 1454)**, **/PCIRCLE (p. 1370)**, and **/PWEDGE (p. 1557)** commands.

The command is generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if annotation is used. It is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LSREAD, *LSNUM*

Reads load and load step option data into the database.

SOLUTION (p. 38): Load Step Operations (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSNUM

Identification number of the load step file to be read. Defaults to 1 + highest number read in the current session. Issue **LSREAD (p. 1083)**,STAT to list the current value of *LSNUM*. Issue **LSREAD (p. 1083)**,INIT to reset *LSNUM* to 1. The load step files are assumed to be named Jobname.S*n*, where *n* is a number assigned by the **LSWRITE (p. 1087)** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for *LSNUM* > 99.

Notes

Reads load and load step option data from the load step file into the database. **LSREAD (p. 1083)** *will not* clear the database of *all* current loads. However, if a load is respecified with **LSREAD (p. 1083)**, then it will overwrite the existing load. See the **LSWRITE (p. 1087)** command to write load step files, and the **LSDELE (p. 1073)** command to delete load step files. **LSREAD (p. 1083)** removes any existing **SFGRAD (p. 1772)** specification.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Read LS File

Main Menu>Solution>Load Step Opts>Read LS File

***LSRESTORE**, *EngineName*, *FileName*

Restores a linear solver engine from a binary file.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

EngineName

Name used to identify this engine.

FileName

Name of the file to read from.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Restores a previously dumped Linear Solver (see the ***LSDUMP (p. 1073)** command). This Linear Solver can be used to solve a linear system using the ***LSBAC (p. 1066)** command.

Menu Paths

This command cannot be accessed from a menu.

LSSCALE, *NL1*, *NL2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of lines from a pattern of lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1*, *NL2*, *NINC

Set of lines (*NL1* to *NL2* in steps of *NINC*) that defines the pattern to be scaled. *NL2* defaults to *NL1*, *NINC* defaults to 1. If *NL1* = ALL, *NL2* and *NINC* are ignored and the pattern is defined by all selected lines. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

RX*, *RY*, *RZ

Scale factors to be applied to the X, Y, Z keypoint coordinates in active coordinate system (*RR*, *R*θ, *RZ* for cylindrical; *RR*, *R*θ, *R*Φ for spherical). Note that the *R*θ and *R*Φ scale factors are interpreted as angular *offsets*. For example, for **CSYS (p. 441)**,1, *RR*, *R*θ, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Nodes and line elements associated with the original lines will be generated (scaled) if they exist.

1

Nodes and line elements will *not* be generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IMOVE

Specifies whether lines will be moved or newly defined:

0

Additional lines will be generated.

1

Original lines will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

This command generates a scaled set of lines (and their corresponding keypoints and mesh) from a pattern of lines.

The MAT, TYPE, REAL, and ESYS attributes are based on the lines in the pattern and not the current settings.

Scaling occurs in the active coordinate system. Lines in the pattern could have been generated in any coordinate system.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Lines

LSSOLVE, *LSMIN*, *LSMAX*, *LSINC*

Reads and solves multiple load steps.

SOLUTION (p. 38): Load Step Operations (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | PrepPost | **Solver** | AS add-on

LSMIN*, *LSMAX*, *LSINC

Range of load step files to be read and solved, from *LSMIN* to *LSMAX* in steps of *LSINC*. *LSMAX* defaults to *LSMIN*, and *LSINC* defaults to 1. If *LSMIN* is blank, a brief command description is displayed. The load step files are assumed to be named `Jobname.S`*n*, where *n* is a number assigned by the **LSWRITE (p. 1087)** command (01--09,10,11, etc.). On systems with a 3-character limit on the extension, the "S" is dropped for numbers > 99.

Notes

This command invokes a Mechanical APDL macro (LSSOLVE.MAC) to read and solve multiple load steps.

The macro loops through a series of load step files written by the **LSWRITE (p. 1087)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command cannot be used with the birth-death option, does not support cyclic symmetry analysis, and does not support restarts.

Menu Paths

Main Menu>Solution>Solve>From LS Files

LSTR, *P1*, *P2*

Defines a straight line irrespective of the active coordinate system.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1

Keypoint at the beginning of line. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI)

P2

Keypoint at the end of line.

Notes

Defines a straight line from *P1* to *P2* using the global Cartesian coordinate system. The active coordinate system will be ignored. The line shape is invariant with the coordinate system after it is generated. Lines may be redefined only if not yet attached to an area.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Straight Line

---

LSUM

Calculates and prints geometry statistics of the selected lines.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Calculates and prints geometry statistics (length, centroid, moments of inertia, etc.) associated with the selected lines. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed, irrespective of any material associations (**LATT (p. 1014)**, **MAT (p. 1111)**). Items calculated by **LSUM (p. 1086)** and later retrieved by a ***GET (p. 797)** or ***VGET (p. 2118)** command are valid only if the model is not modified after the **LSUM (p. 1086)** command is issued.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Lines

---

LSWRITE, *LSNUM*

Writes load and load step option data to a file.

SOLUTION (p. 38): Load Step Operations (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSNUM

Number to be assigned to the load step file name for identification purposes. Defaults to 1 + highest *LSNUM* used in the current session. Issue **LSWRITE (p. 1087)**,STAT to list the current value of *LSNUM*. Issue **LSWRITE (p. 1087)**,INIT to reset to 1. The load step file will be named `Jobname.S`*n*, where *n* is the specified *LSNUM* value (preceded by "0" for values 1-9). On systems with a 3-character limit on the file name extension, the "S" is dropped for *LSNUM* > 99.

Notes

Writes all load and load step option data for the selected model to a load step file for later use. **LSWRITE (p. 1087)** does *not* capture changes made to real constants (**R (p. 1565)**), material properties (**MP (p. 1160)**), couplings (**CP (p. 421)**), or constraint equations (**CE (p. 297)**).

Solid model loads will not be saved if the model is not meshed. Solid model loads, if any, are transferred to the finite element model. Issue **LSCLEAR (p. 1071)**,FE to delete finite element loads.

One file is written for each load step. Use the **LSREAD (p. 1083)** command to read a single load step file, and the **LSDELE (p. 1073)** command to delete load step files. Use the **LSSOLVE (p. 1085)** command to read and solve the load steps sequentially.

Solution control commands are typically not written to the file unless you specifically change a default solution setting.

LSWRITE (p. 1087) does not support the following commands: **DJ (p. 537)**, **FJ (p. 759)**, **GSBDATA (p. 876)**, **GSGDATA (p. 877)**, **ESTIF (p. 681)**, **EKILL (p. 616)**, **EALIVE (p. 592)**, **MPCHG (p. 1167)**, and **OUTRES (p. 1336)**. These commands will not be written to the load step file.

LSWRITE (p. 1087) cannot be used with the birth-death option.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Write LS File

Main Menu>Solution>Load Step Opts>Write LS File

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/LSYMBOL, *X*, *Y*, *SYMANG*, *SYMTYP*, *SYMSIZ*, *KEYBMP*

Creates annotation symbols (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X

X location for symbol (-1.0 < X < 2.0).

Y

Y location for symbol (-1.0 < Y < 1.0).

SYMANG

Symbol orientation angle.

SYMTYP

Symbol type:

1

Arrow.

2

Tee.

3

Circle.

4

Triangle.

5

Star.

SYMSIZ

Symbol size multiplier (0.1 to 20.0). Defaults to 1.0.

KEYBMP

If *KEYBMP* = 1, the annotation is a bitmap. *SYMTYP* will then be a number from 1-99, indicating the bitmap type (see notes), and *X* and *Y* will define the lower left corner of the bitmap. The *SYMANG*, *SYMSIZ*arguments are ignored. If *KEYBMP* = 0, or blank, then the argument definitions above apply.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines annotation symbols to be written directly onto the display at a specified location. This is a command generated by the GUI and will appear in the log file (`Jobname.log`) if annotation is used. This command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

All symbols are shown on subsequent displays unless the annotation is turned off or deleted. Use the **/LSPEC (p. 1081)** command to set the attributes of the symbol.

The *KEYBMP* argument reads the symtype argument to determine which bitmap to insert. This bitmap is defined by an integer between 1 and 99. Numbers 1 through 40 correspond to the standard texture values found in the **/TXTRE (p. 2029)** command, while numbers 51 through 99 correspond to user supplied bitmaps, as defined using the Filename option of the **/TXTRE (p. 2029)** command. Numbers 51 through 57 are predefined (the logos, clamps and arrows available from the GUI) but can be overridden. Numbers 41 through 50 are reserved.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

LSYMM, *Ncomp*, *NL1*, *NL2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Generates lines from a line pattern by symmetry reflection.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ncomp

Symmetry key:

X

X symmetry (default).

Y

Y symmetry.

Z

Z symmetry.

NL1*, *NL2*, *NINC

Reflect lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines (**LSEL (p. 1074)**). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Generate nodes and elements associated with the original lines, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether areas will be moved or newly defined:

0

Generate additional lines.

1

Move original lines to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of lines (and their corresponding keypoints and mesh) from a given line pattern by a symmetry reflection (see analogous node symmetry command, **NSYM (p. 1289)**). The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be Cartesian. Lines in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Lines are generated as described in the **LGEN (p. 1046)** command.

See the **ESYM (p. 684)** command for additional information about symmetry elements.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LTAN, *NL1*, *P3*, *XV3*, *YV3*, *ZV3*

Generates a line at the end of, and tangent to, an existing line.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NL1

Number of the line the generated line is tangent to. If negative, assume *P1* (see below), instead of *P2*, is the second keypoint of line *NL1*. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P3

Keypoint at which generated line must end.

The following fields are used only if a specified slope on the end of the new line is desired, otherwise a zero curvature end slope will be automatically calculated. To specify the end slope, use the following fields to define a "slope vector" that has its tail at the origin and its head at the point *XV*, *YV*, *ZV* in the active coordinate system (**CSYS (p. 441)**). The corresponding end slope of the line will then be parallel to this "slope vector."

XV3, YV3, ZV3

Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P3* end of the line. The tail of the vector is at the coordinate system origin.

Notes

Generates a line (*P2-P3*) tangent at end point (*P2*) of line *NL1* (*P1-P2*).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Lines>Tangent to Line

LTRAN, *KCNTO*, *NL1*, *NL2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of lines to another coordinate system.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCNTO

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NL1, NL2, NINC

Transfer lines from pattern beginning with *NL1* to *NL2* (defaults to *NL1*) in steps of *NINC* (defaults to 1). If *NL1* = ALL, *NL2* and *NINC* are ignored and pattern is all selected lines (**LSEL (p. 1074)**). If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NL1* (*NL2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Generate nodes and elements associated with the original lines, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether lines will be moved or newly defined:

0

Generate additional lines.

1

Move original lines to new position retaining the same keypoint numbers (*KINC* and *NOELM* are ignored). Valid only if the old lines are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of lines (and their corresponding keypoints and mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER (p. 2011)**). The MAT, TYPE, REAL, and ESYS attributes are based upon the lines in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Lines are generated as described in the **LGEN (p. 1046)** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LUMPM, *Key*, - -, *KeyElt*

Specifies a lumped mass matrix formulation.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Formulation key:

OFF

Use the element-dependent default mass matrix formulation (default).

ON

Use a lumped mass approximation.

- -

Unused field.

KeyElt

Formulation key for elements with rotational degrees of freedom; applicable only when the lumped mass formulation key is turned on (*Key* = ON):

0 (blank)

Use direct diagonalization of the element mass matrix (default).

1

Use translational mass only.

2

Use the frame invariant formulation.

Command Default

Use the default element mass matrix.

Notes

In a modal analysis, the lumped mass matrix option (**LUMPM (p. 1093)**,ON) is not allowed when using the Supernode mode-extraction method (**MODOPT (p. 1140)**,SNODE). The eigensolver will automatically be switched to Block Lanczos (LANB) in this case.

In the use pass of a substructuring analysis, the lumped mass matrix formulation (**LUMPM (p. 1093)**,ON) modifies the superelement mass matrix and may give unexpected results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The translational mass only option (**LUMPM (p. 1093)**,ON,,1) applies to the following elements: SHELL181, BEAM188, BEAM189, SHELL208, SHELL209, SHELL281, PIPE288, PIPE289, and ELBOW290. The frame invariant formulation (**LUMPM (p. 1093)**,ON,,2) applies only to BEAM188, BEAM189, PIPE288, and PIPE289 elements.

For more information, see Lumped Matrices in the *Theory Reference*.

This command is also valid in PREP7. If used in SOLUTION, this command is valid only within the first load step.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Analysis Type>New Analysis

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>New Analysis

LVSCALE, *FACT*, *LDSTEP*

Scales the load vector for mode-superposition analyses.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FACT

Scale factor applied to both the real and imaginary (if they exist) components of the load vector. Defaults to 0.0.

LDSTEP

Specifies the load step number from the modal analysis (**MODCONT (p. 1132)**,ON). It corresponds to the load vector number. Defaults to 1. The maximum admissible value is the number of vectors written in the `Jobname.mode` file.

Command Default

No load vector is applied.

Notes

Specifies the scale factor for the load vector that was created in a modal (**ANTYPE (p. 162)**,MODAL) analysis. Applies only to the mode-superposition transient analysis (**ANTYPE (p. 162)**,TRANS), mode-superposition harmonic analysis (**ANTYPE (p. 162)**,HARMIC), random vibration analysis (**ANTYPE (p. 162)**,SPECTR with **SPOPT (p. 1837)**,PSD), and multiple point response spectrum analysis (**ANTYPE (p. 162)**,SPECTR with **SPOPT (p. 1837)**,MPRS). For PSD and MPRS analyses, **LVSCALE (p. 1094)** is only applicable for pressure loading.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **LVSCALE (p. 1094)** command supports tabular boundary conditions (%TABNAME_X%) for *FACT* input values only as a function of time in the mode-superposition transient (**ANTYPE (p. 162)**,TRANS) or as a function of frequency in mode-superposition harmonic (**ANTYPE (p. 162)**,HARMIC).

MPC contact generates constraint equations that can include constant terms (included on the right-hand side of the system equation). The **LVSCALE (p. 1094)** command scales the constant terms.

In mode-superposition transient and harmonic analyses, all of the load vectors need to be scaled in the first load step. Use a zero scale factor if they are not actually used in this first load step. Similarly, in random vibration and multipoint response spectrum analyses, all of the load vectors need to be scaled in the first participation factor calculation (**PFACT (p. 1385)**). Use a zero scale factor if they are not actually used for the first input table.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For Mode Super

Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For PSD

Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For Mode Super

Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For PSD

Main Menu>Solution>Define Loads>Apply>Load Vector>For Mode Super

Main Menu>Solution>Define Loads>Apply>Load Vector>For PSD

Main Menu>Solution>Define Loads>Delete>Load Vector>For Mode Super

Main Menu>Solution>Define Loads>Delete>Load Vector>For PSD

LWPLAN, *WN*, *NL1*, *RATIO*

Defines the working plane normal to a location on a line.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified.

NL1

Number of line to be used. If *NL1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

RATIO

Location on *NL1*, specified as a ratio of the line length. Must be between 0.0 and 1.0. If *RATIO* = P, use graphical picking to specify location on the line.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Working plane is parallel to the Global X-Y plane at Z = 0.0.

Notes

Defines a working plane (to assist in picking operations) normal to a location on a line. See **WPSTYL (p. 2182)** command to set the style of working plane display.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Align WP with>Plane Normal to Line

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

M Commands

M, *NODE*, *Lab1*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *SUPPORT*

Defines master degrees of freedom (MDOFs) for superelement generation analyses.

SOLUTION (p. 38): Master DOF (p. 46)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node number at which an MDOF is defined. If ALL, define MDOFs at all selected nodes (**NSEL (p. 1266)**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab1

Valid degree-of-freedom label. If ALL, use all appropriate labels. *Structural labels*: UX, UY, or UZ (displacements); ROTX, ROTY, or ROTZ (rotations). *Thermal labels*: TEMP, TBOT, TE2, TE3, . . ., TTOP (temperature). *Electric labels*: VOLT (voltage).

NEND, NINC

Define all nodes from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1) as MDOFs in the specified direction.

Lab2, Lab3, Lab4, . . . , Lab6

Additional MDOF labels. The nodes defined are associated with each label specified.

SUPPORT

Pseudo-constraints key for the free-interface (**CMSOPT (p. 378)**,FREE) and residual-flexible free-interface (**CMSOPT (p. 378)**, RFFB) CMS method analyses:

OFF – Defined MDOFs remain free during the mode-extraction analysis (default).

ON – Defined MDOFs are constrained during the mode-extraction analysis.

Notes

Defines master degrees of freedom (MDOFs) for superelement generation. If defined for other analyses, MDOFs are ignored. If used in the SOLUTION processor, this command is valid only within the first load step.

Reissue **M (p. 1097)** for additional MDOFs. The number of master nodes allowed is limited only by the maximum system memory available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The substructure (**ANTYPE (p. 162)**,SUBSTR) analysis uses the matrix condensation technique to reduce the structure matrices to those characterized by a set of MDOFs.

MDOFs are identified by a list of nodes and their nodal directions. The actual degree-of-freedom directions available for a given node depends upon the degrees of freedom associated with element types (**ET (p. 686)**) at that node.

There must be some mass (or stress stiffening in the case of a buckling analysis) associated with each MDOF (except for the VOLT label). The mass may be due either to the distributed mass of the element or due to discrete lumped masses at the node.

If an MDOF is specified at a constrained point, it is ignored.

If an MDOF is specified at a coupled node, it should be specified at the prime node of the coupled set.

For cyclic symmetry superelements, if MDOFs are defined at both low- and high-edge nodes, the cyclic constraint equations between those nodes are ignored.

Substructure analysis connection points must be defined as MDOFs.

The *SUPP0RT* argument is ignored for the fixed-interface CMS method analysis (**CMSOPT (p. 378)**,FIX).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>Define

Main Menu>Preprocessor>Modeling>CMS>CMS Interface>Define

Main Menu>Solution>Master DOFs>Define

MACOPT,*Option*,*Value1*,*Value2*,*Value3*

Specifies modal assurance criterion (MAC) or frequency response function (FRF) correlation criteria calculation options for RSTMAC (p. 1627).

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Matching / Mapping options:		
Option Label	**Description**	***Value1*, *Value2*, *Value3***
(blank)	Matching method based on the node location (default).	NA
NUMMATCH	Key to activate the node matching method based on the node number.	*Value1*: 0 (OFF or NO) – Node number matching is off, and nodes are matched based on node location. Tolerance for node matching may also be applied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		1 (ON or YES) – Nodes are matched based on node number and solutions are compared at matched nodes only.
NODMAP	Key to activate node mapping.	*Value1*: 0 (OFF or NO) – Node mapping is off, and nodes are matched based on node location. Tolerance for node matching may also be applied. 1 (ON or YES) – Nodes are mapped and solution is interpolated.
Options for matching by location (not applicable to NUMMATCH and NODMAP methods):		
ABSTOLN	Absolute tolerance for node matching. Relative tolerance setting (RELTOLN) is ignored. This is the default option.	*Value1*: Absolute tolerance value. Defaults to 0.01.
RELTOLN	Relative tolerance for node matching. Absolute tolerance setting (ABSTOLN) is ignored. Relative tolerance is based on a fraction of the minimum element dimension.	*Value1*: Relative tolerance value. It must be greater than 0 and less than or equal to 1. Defaults to 0.5 (50%).
KEYALGONOD	Key to specify the algorithm for node matching based on location.	*Value1*: 0 (OFF or NO) – The first node with distance below tolerance is considered a match (default). 1 (ON or YES) – The nearest node with distance below tolerance is considered a match.
MAC calculation options :		
DOF	Key to specify the degrees of freedom used. Defaults to all structural degrees of freedom. DOF selection is not supported with node mapping.	*Value1*: Supported labels are: UX, UY, UZ, ROTX, ROTY, ROTZ, U (all translations), ROT (all rotations), STRU (all structural DOFs) PRES, VOLT, CONC, MAG, TEMP, AZ *Value2*: (blank) – Select DOF DELE – Reset to default (all structural DOFs)
KEYMASS	Key to include the mass matrix diagonal in the MAC calculations.	*Value1*:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		0 (OFF or NO) – Do not include the mass. 1 (ON or YES) – Include the mass if available (default).
NMAC	Key to activate the computation of MAC per node pair	*Value1*: 0 (OFF or NO) – Do not activate the node pair MAC calculation (default). 1 (ON or YES) – Activate the node pair MAC calculation.
UNV file options :		
FRF (p. 1102)	Key to activate the frequency response function (FRF) correlation method and specify required inputs. Only applies when File2 on **RSTMAC (p. 1627)** is a universal format file (.unv) and contains Dataset 58.	*Value1*: Key to activate/deactivate FRF correlation methods and specify the needed criteria: YES (or blank) – Activates the cross signature assurance criterion (default). ALL – Activates the computation of both the cross signature assurance criterion and the cross signature scale factor. NO – Deactivates the FRF correlation and activates the default MAC calculations. *Value2* and *Value3* are ignored. *Value2*: Key to specify the considered response degree of freedom in the .rst file. The following options are available: 1– +X Translation 4– +X Rotation 2– +Y Translation 5– +Y Rotation 3– +Z Translation 6– +Z Rotation *Value3*: Key to specify the considered response degree of freedom in the .unv file. The following options are available: 1 – +X Translation 4– +X Rotation -1 – -X Translation -4– -X Rotation 2 – +Y Translation 5– +Y Rotation -2 – -Y Translation -5– -Y Rotation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		3 – +Z Translation 6– +Z Rotation -3 – -Z Translation -6– -Z Rotation
UNVSCALE	Scaling of the nodal coordinates when *File2* on **RSTMAC (p. 1627)** is a universal format file (.unv).	*Value1*: Blank or 0 – No scaling (default). Non-zero value – Scaling factor applied to nodal coordinates.
UNVTRAN	Key to control the transformation of the solution vector from the local to the global coordinate system. Only applies when *File2* on **RSTMAC (p. 1627)** is a universal format file (.unv) and contains Dataset Number 2420.	*Value1*: 0 (OFF or NO) – No transformation is performed (default). 1 (ON or YES) – Transformation of the solution vector is performed.
UNVDEBUG	Key to control the printout of the universal format file records. All information is listed on file *File2_unv.txt*. Only applies when *File2* on **RSTMAC (p. 1627)** is a universal format file (.unv).	*Value1*: 0 (OFF or NO) – No printout (default). 1 (ON or YES) – Print file records.

Command Default

If **MACOPT (p. 1098)** is not issued prior to **RSTMAC (p. 1627)**, node matching based on location is used by default in MAC calculations performed by **RSTMAC (p. 1627)**. Unless otherwise specified, an absolute tolerance (ABSTOLN) value of 0.01 is used for the node matching.

Notes

The **RSTMAC (p. 1627)** command calculates the MAC or FRF criteria values based on the options specified via **MACOPT (p. 1098)**. The **MACOPT (p. 1098)** command must be issued before the **RSTMAC (p. 1627)** command. These commands enable you to compare nodal solutions from two results files (.rst or .rstp) or from one results file and one Universal Format file (.unv). Multiple **MACOPT (p. 1098)** commands can be issued to specify which results are compared and how.

As listed in the table above, model solutions can be compared using three different mehtods: matching nodes based on location, matching nodes based on node number, or by node mapping and solution interpolation.

When node mapping and solution interpolation is performed (*Option* = NODMAP), the following applies:

- *File1* on **RSTMAC (p. 1627)** must correspond to a model meshed in solid and/or shell elements. Other types of elements can be present, but the node mapping is not performed for these elements.
- You should only compare solutions of models having the same dimension (both models are 2D or both models are 3D). Comparing models with different dimensions may lead to incorrect results if the solution at mapped/matched nodes is not representative of the global solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Interpolation is performed on UX, UY, and UZ degrees of freedom.

Node pair MAC computation (*Option* = NMAC) is only supported when a matching procedure is used and a specific substep number is requested for each solution (*Sbstep1*, *Sbstep2* on **RSTMAC (p. 1627)** command).

Non-structural degrees of freedom in coupled-field analyses are supported for the matching methods (*Option* = ABSTOLN or NUMMATCH). Multiple **MACOPT (p. 1098)**, DOF commands can be issued consecutively to combine different degrees of freedom.

The FRF option is not compatible with any MAC calculation options (DOF, NMAC and KEYMASS) and can only be used with node matching procedures.

Example Usage

For a detailed discussion on using **MACOPT (p. 1098)** with examples, see Comparing Nodal Solutions From Two Models or From One Model and Experimental Data (RSTMAC) in the *Basic Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

MAGOPT, *Value*

Specifies options for a 3D magnetostatic field analysis.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Value

Option key:

0

Calculate a complete H field solution in the entire domain using a single (reduced) potential.

> **Caution:**
>
> When used in problems with both current sources and iron regions, errors may result due to numerical cancellation.

1

Calculate and store a preliminary H field in "iron" regions ($\mu_r \neq 1$). Requires flux-parallel boundary conditions to be specified on exterior iron boundaries. Used in conjunction with subsequent solutions with *VALUE* = 2 followed by *VALUE* = 3. Applicable to multiply-connected iron domain problems.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2

Calculate and store a preliminary H field in "air" regions (μ_r = 1). The air-iron interface is appropriately treated internally by the program. Used in conjunction with a subsequent solution with *VALUE* = 3. Applicable to singly-connected iron domain problems (with subsequent solution with *VALUE* = 3) or to multiply-connected iron domain problems (when preceded by a solution with *VALUE* = 1 and followed by a solution with *VALUE* = 3).

3

Use the previously stored H field solution(s) and calculate the complete H field.

Notes

Specifies the solution sequence options for a 3D magnetostatic field analysis using a scalar potential (MAG). The solution sequence is determined by the nature of the problem.

You cannot use constraint equations with *Value* = 1.

This command is also valid in PREP7.

Distributed-Memory Parallel (DMP) Restriction

MAGOPT (p. 1102),3 is not supported in a DMP solution when the following contact elements are present in the model: CONTA174, CONTA175, or CONTA177.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>DSP Method

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>GSP Method

Main Menu>Preprocessor>Loads>Load Step Opts>Magnetics>Options Only>RSP Method

Main Menu>Solution>Load Step Opts>Magnetics>Options Only>DSP Method

Main Menu>Solution>Load Step Opts>Magnetics>Options Only>GSP Method

Main Menu>Solution>Load Step Opts>Magnetics>Options Only>RSP Method

---

MAGSOLV, *OPT*, *NRAMP*, *CNVCSG*, *CNVFLUX*, *NEQIT*, *BIOT*, *CNVTOL*

Specifies magnetic solution options and initiates the solution.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

OPT

Static magnetic solution option:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Vector potential (MVP) or edge formulation (default).

1

Combined vector potential and reduced scalar potential (MVP-RSP).

2

Reduced scalar potential (RSP).

3

Difference scalar potential (DSP).

4

General scalar potential (GSP).

NRAMP

Number of ramped substeps for the first load step of a nonlinear MVP or MVP-RSP solution. Defaults to 3. If *NRAMP* = -1, ignore the ramped load step entirely.*NRAMP* is ignored for linear magnetostatics.

CNVCSG

Tolerance value on the program-calculated reference value for the magnetic current-segment convergence. Used for the MVP, the MVP-RSP, and the edge formulation solution options (*OPT* = 0 and 1). Defaults to 0.001.

CNVFLUX

Tolerance value on the program-calculated reference value for the magnetic flux convergence. Used for all scalar potential solution options (*OPT* = 2, 3, 4). Defaults to 0.001.

NEQIT

Maximum number of equilibrium iterations per load step. Defaults to 25.

BIOT

Option to force execution of a Biot-Savart integral solution (**BIOT (p. 265)**,NEW) for the scalar potential options. Required if multiple load steps are being performed with different current source primitives (SOURC36 elements).

0

Do not force execution of Biot-Savart calculation (default); Biot-Savart is automatically calculated only for the first solution.

1

Force execution of Biot-Savart calculation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CNVTOL

Sets the convergence tolerance for AMPS reaction. Defaults to 1e-3.

Notes

MAGSOLV (p. 1103) invokes a Mechanical APDL macro which specifies magnetic solution options and initiates the solution. The macro is applicable to any Mechanical APDL magnetostatic analysis using the magnetic vector potential (MVP), reduced scalar potential (RSP), difference scalar potential (DSP), general scalar potential (GSP), or combined MVP-RSP formulation options. Results are only stored for the final converged solution. (In POST1, issue ***SET (p. 1720)**,LIST to identify the load step of solution results.) The macro internally determines if a nonlinear analysis is required based on magnetic material properties.

If you use the *BIOT* option and issue **SAVE (p. 1645)** after solution or postprocessing, the Biot-Savart calculations are saved to the database, but will be overwritten upon normal exit from the program. To save this data after issuing **SAVE (p. 1645)**, use the **/EXIT (p. 708)**,NOSAVE command. You can also issue the **/EXIT (p. 708)**,SOLU command to exit Mechanical APDL and save all solution data, including the Biot-Savart calculations, in the database. Otherwise, when you issue **RESUME (p. 1601)**, the Biot-Savart calculation will be lost (resulting in a zero solution).

The MVP, MVP-RSP, and edge formulation options perform a two-load-step solution sequence. The first load step ramps the applied loads over a prescribed number of substeps (*NRAMP*), and the second load step calculates the converged solution. For linear problems, only a single load step solution is performed. The ramped load step can be bypassed by setting *NRAMP* to -1.

The RSP option solves in a single load step using the adaptive descent procedure. The DSP option uses two load steps, and the GSP solution uses three load steps.

The following analysis options and nonlinear options are controlled by this macro: **KBC (p. 970)**, **NEQIT (p. 1214)**, **NSUBST (p. 1287)**, **CNVTOL (p. 396)**, **NROPT (p. 1261)**, **MAGOPT (p. 1102)**, and **OUTRES (p. 1336)**.

You cannot use constraint equations with *OPT* = 4.

When the *BIOT* option is on (*BIOT* = 1), Distributed-Memory Parallel (DMP) restrictions may apply. For more information, see the **BIOT (p. 265)** command.

Menu Paths

Main Menu>Solution>Solve>Electromagnet>Static Analysis>Opt&Solv

---

MAP, - - , *kDIM*, - - , *kOUT*, *LIMIT*

Maps pressures from source points to target surface elements.

/MAP (p. 63): Pressure Mapping (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

- -

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

kDIM

Interpolation key:

0 or 2

Interpolation is done on a surface (default).

3

Interpolation is done within a volume. This option is useful if the supplied source data is volumetric field data rather than surface data.

--

Unused field.

kOUT

Key to control how pressure is applied when a target node is outside of the source region:

0

Use the pressure(s) of the nearest source point for target nodes outside of the region (default).

1

Set pressures outside of the region to zero.

LIMIT

Number of nearby points considered for interpolation. The minimum is 5; the default is 20. Lower values reduce processing time. However, some distorted or irregular meshes will require a higher *LIMIT* value to find the points encompassing the target node in order to define the region for interpolation.

Notes

Maps pressures from source points to target surface elements.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/MAP

Enters the mapping processor.

/MAP (p. 63): Pressure Mapping (p. 63)
SESSION (p. 11): Processor Entry (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the mapping processor. This processor is used to read in source data from an external file and map it to the existing geometry.

The current database is saved (to `BeforeMapping.db`) upon entering the processor, and it is resumed upon exiting (**FINISH (p. 758)** command). Any nodes or elements not on the target surface are deleted for easier viewing of the mapping quantities. A database of this mapping geometry (`Mapping.db`) is also saved at the **FINISH (p. 758)** command.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

MAP2DTO3D,*Action*,*VALUE1*,*VALUE2*

Initiates a 2D to 3D analysis and maps variables.

SOLUTION (p. 38): 2D to 3D Analysis (p. 47)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

The 2D to 3D analysis action to perform:

START

Start the analysis process by rebuilding the 2D analysis database (`.db`) based on the specified load step and substep information, and update nodes to their deformed positions in the 2D mesh.

VALUE1

The load step number at which 2D to 3D analysis should occur. The default value is the highest load step number found in the `Jobname.R`*nnn* files (for the current jobname and in the current directory).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VALUE2

The substep number of the specified load step (*VALUE1*) at which the 2D to 3D analysis should occur. The default value is the highest substep number found in the specified load step in the `Jobname.Rnnn` files (for the current jobname and in the current directory).

FINISH

Maps boundary conditions and loads from the 2D mesh to the extruded 3D mesh. (*VALUE1* and *VALUE2* are not used.)

SOLVE

Map nodal and element solutions from 2D to 3D and rebalance the results.

VALUE1

The maximum number of substeps allowed during rebalancing. Default = 500.

Notes

The **MAP2DTO3D (p. 1107)** command initiates the 2D to 3D analysis process, sets analysis options, rebuilds the database, and maps the solution variables from the 2D mesh to the 3D mesh.

Before issuing this command, clear the database (**/CLEAR (p. 350)**).

For more information, see 2D-to-3D Analysis in the *Advanced Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

MAPSOLVE, *MAXSBSTEP*

Maps solved node and element solutions from an original mesh to a new mesh.

SOLUTION (p. 38): Rezoning (p. 46)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAXSBSTEP

The maximum number of substeps for rebalancing the residuals. The default value is 5.

Notes

Used during the rezoning process, the **MAPSOLVE (p. 1108)** command maps solved node and element solutions from the original mesh to the new mesh and achieves equilibrium based on the new mesh.

Additional substeps are necessary to reduce the residuals to zero.

During the rebalancing stage, the external loads and time remain unchanged.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **MAPSOLVE (p. 1108)** command is valid only for rezoning (**REZONE (p. 1606)**).

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

Main Menu>Solution>Manual Rezoning>Map Results

MAPVAR,*Option*,*MatId*,*IstrtStress*,*nTenStress*,*IstrtStrain*,*nTen-Strain*,,*IstrtVect*,*nVect*

Defines tensors and vectors in user-defined state variables for rezoning and in 2D to 3D analyses.

SOLUTION (p. 38): Rezoning (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

DEFINE

Define variables for the specified *MatId* material ID (default).

LIST

List the defined variables for the specified *MatId* material ID.

MatId

The material ID for the state variables which you are defining (*Option* = DEFINE) or listing (*Option* = LIST).

When *Option* = LIST, the default value for this argument is ALL (which lists *all* defined variables). When *Option* = DEFINE, you must explicitly specify a material ID.

IstrtStress

The start position of stress-like tensors in the state variables. This value must be either a positive integer or 0 (meaning no stress-like tensors).

nTenStress

The number of stress-like tensors in the state variables. This value must be either a positive integer (or 0), and all stress-like tensors must be contiguous.

IstrtStrain

The start position of strain-like tensors in the state variables. This value must be either a positive integer or 0 (meaning no strain-like tensors).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nTenStrain

The number of strain-like tensors in the state variables. This value must be either a positive integer (or 0), and all strain-like tensors must be contiguous.

IstrtVect

The start position of vectors in the state variables. This value must be either a positive integer or 0 (meaning no vectors).

nVect

The number of vectors in the state variables. This value must be either a positive integer (or 0), and all vectors must be contiguous.

Notes

The **MAPVAR (p. 1109)** command identifies the tensors and vectors in user-defined state variables (**TB (p. 1899)**,STATE) for user-defined materials (**TB (p. 1899)**,USER and `UserMat` or `UserMatTh`) or user-defined creep laws (**TB (p. 1899)**,CREEP,,,,100 and `UserCreep`).

To handle large-rotation effects and to correctly differentiate between tensor- and vector-mapping, specify the start position of specific state variables. For stress-like tensors, the shear components saved as state variables are the tensor component. For strain-like tensors, the shear components saved as state variables are twice the tensor components. Therefore, issue the **MAPVAR (p. 1109)** command to define the stress-like and strain-like tensors individually. The command ensures that user-defined state variables are mapped correctly during rezoning and in 2D to 3D analyses.

In a rezoning operation, **MAPVAR (p. 1109)** must be issued *after* remeshing (**REMESH (p. 1581)**,FINISH) but *before* mapping (**MAPSOLVE (p. 1108)**).

Menu Paths

This command cannot be accessed from a menu.

MASCALE,*massFact*

Activates scaling of the entire system matrix.

`SOLUTION` (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

massFact

Scaling factor (> 0) for the mass matrix. Default = 1.0.

Command Default

No scaling is applied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is supported in the first load step of the analysis only.

The following features are not affected by the scaling:

- Ocean loading (Applying Ocean Loading from a Hydrodynamic Analysis)
- Steady-state rolling (**SSTATE (p. 1855)**)

The mass-related information (mass, center of mass, and mass moments of inertia) printed in the mass summary is based on unscaled mass properties.

Menu Paths

This command cannot be accessed from a menu.

MASTER

Specifies "Master DOF" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Master DOF

MAT, *MAT*

Sets the element material attribute pointer.

PREP7 (p. 22): Meshing (p. 29)
PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Assign this material number to subsequently defined elements (defaults to 1).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Identifies the material number to be assigned to subsequently defined elements. This number refers to the material number (*MAT*) defined with the material properties (**MP (p. 1160)**). Material numbers may be displayed (**/PNUM (p. 1457)**).

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

MATER

Specifies "Material properties" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Materials

MAXCYCMODE, *LSTEP1*, *SBSTEP1*, *Cmp1x1*, *LSTEP2*, *SBSTEP2*, *Cmp1x2*, *Comp*, *Numsect*

Computes the maximum results value of cyclic mode pairs at the same frequency or the amplitude of a single complex mode.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSTEP1

Load step for the first mode in the pair. (Set *LSTEP1* and *LSTEP2* to the same integer value if you are using **MAXCYCMODE (p. 1112)** to calculate the amplitude of a single complex mode.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SBSTEP1

Substep for the first mode in the pair. (Set *SBSTEP1* and *SBSTEP2* to the same integer value if you are using **MAXCYCMODE (p. 1112)** to calculate the amplitude of a single complex mode.)

Cmp1x1

Complex value key for the first mode in the pair.

'Real'

Indicates the real part of the first mode (default).

'Imag'

Indicates the imaginary part of a complex mode. (Used only if you are using **MAXCYCMODE (p. 1112)** to calculate the amplitude of a single complex mode.)

LSTEP2

Load step for the second mode in the pair. (Set *LSTEP1* and *LSTEP2* to the same integer value if you are using MAXCYCMODE to calculate the amplitude of a single complex mode.)

SBSTEP2

Substep for the second mode in the pair. (Set *SBSTEP1* and *SBSTEP2* to the same integer value if you are using MAXCYCMODE to calculate the amplitude of a single complex mode.)

Cmp1x2

Complex value key for the second mode in the pair.

'Real'

Indicates the real part of the second mode (default).

'Imag'

Indicates the imaginary part of a complex mode. (Used only if you are using **MAXCYCMODE (p. 1112)** to calculate the amplitude of a single complex mode.)

Comp

Component for which a maximum value is to be obtained. Valid *Comp* labels and component descriptions are listed in the Notes (p. 1114) section.

Numsect

Number of sectors. This should only be used for cyclically symmetric modes computed using the **CYCLIC (p. 462)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

There is no command default (p. 7), and only *Cmp1x1* and *Cmp1x2* have argument defaults (p. 3). Values must be specified for all other command arguments except for *Numsect*, which is only required if the command is used for computing the maximum of cyclic modes calculated in a cyclic symmetry analysis using the **CYCLIC (p. 462)** command.

This command is valid only for mass normalized modes (**MODOPT (p. 1140)**,,,,,,OFF).

For a model generated with the **CYCLIC (p. 462)** command, results are only valid when the solution coordinate system is aligned with the cyclic coordinate system. Issuing **MAXCYCMODE (p. 1112)** while the cyclic sector is expanded (**/CYCEXPAND (p. 454)**,,ON) will result in resetting the cyclic expansion back to the sector (**/CYCEXPAND (p. 454)**,,OFF).

This command has two different functionalities:

- To compute the maximum results value of cyclic mode pairs at the same frequency (p. 1114) that are computed from a cyclically symmetric system.
- To compute the amplitude of a single complex mode (p. 1115) calculated in a modal analysis using the damped eigensolver (**MODOPT (p. 1140)**, DAMP).

To compute the maximum results value of cyclic mode pairs at the same frequency

This command can be used to compute the maximum results value of cyclic mode pairs at the same frequency that are computed from a cyclically symmetric system. Because two modes at the same frequency have indeterminate mode shapes, the combined mode pair must be considered to compare cyclic mode pairs from different solves or models. The maximum values computed are an estimate of the maximum value for each of the modes in the pair.

This command can be used on any mode pairs at the same frequency that are computed in a modal analysis (**ANTYPE (p. 162)**,MODAL) of a cyclically symmetry system modeled in any of the following ways:

- A cyclic symmetry model generated from the **CYCLIC (p. 462)** command
- A cyclic symmetry model generated under the multistage approach using the **MSOPT (p. 1195)** and **CECYCMS (p. 300)** commands
- A cyclic symmetry model generated using no symmetry boundary conditions (full 360° model).

Set the *Cmplx1* and *Cmplx2* values to Real to compute the maximum value of two cyclic modes at the same frequency.

Valid *Comp* labels are listed in the following table. The maximum values obtained are in the coordinate system defined by the **RSYS (p. 1639)** command.

Valid *Comp* Labels	
***Comp* Label**	**Description**
UX	Displacement in the X direction
UY	Displacement in the Y direction

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UZ	Displacement in the Z direction
UCOMP	Displacement in the X, Y, and Z directions
USUM	Displacement vector sum
SX	Stress in the X direction
SY	Stress in the Y direction
SZ	Stress in the Z direction
SCOMP	Stress in the X, Y, and Z directions

To compute the amplitude of a single complex mode

An alternative functionality of this command is to compute the amplitude of a single complex mode that was calculated in a modal analysis using the damped eigensolver (**MODOPT (p. 1140)**, DAMP). In this case, specify the real and imaginary parts of a single mode instead of only real parts of two separate modes. This is useful for cyclic symmetry or multistage cyclic symmetry analyses since **SET (p. 1724)**,,,,AMPL is not supported.

Example Usage

Example 3.4: MAXCYCMODE to Compute the Maximum Results Value of Cyclic Mode Pairs

Menu Paths

This command cannot be accessed from a menu.

MCHECK, *Lab*

Checks mesh connectivity.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Operation:

ESEL

Unselects the valid elements.

Notes

Wherever two area or volume elements share a common face, **MCHECK (p. 1115)** verifies that the way the elements are connected to the face is consistent with their relative normals or integrated volumes. (This may detect folds or otherwise overlapping elements.)

MCHECK (p. 1115) verifies that the element exterior faces form simply-connected closed surfaces. (This may detect unintended cracks in a mesh.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MCHECK (p. 1115) warns if the number of element facets in a 2D loop or 3D shell is not greater than a computed limit. This limit is the smaller of either three times the number of faces on one element, or one-tenth the total number of element faces in the model. (This may detect holes in the middle of a mesh.)

The **MCHECK (p. 1115)** command will perform a number of validity checks on the selected elements, including:

1. **Normal check:** Wherever two area elements share a common edge, **MCHECK (p. 1115)** verifies that the ordering of the nodes on each element is consistent with their relative normals.
2. **Volume check:** Wherever two volume elements share a common face, **MCHECK (p. 1115)** verifies that the sign of the integrated volume of each element is consistent.
3. **Closed surface check:** **MCHECK (p. 1115)** verifies that the element exterior faces form simply-connected closed surfaces (this may detect unintended cracks in a mesh).
4. **Check for holes in the mesh:** **MCHECK (p. 1115)** warns if the number of element faces surrounding an interior void in the mesh is small enough to suggest one or more accidentally omitted elements, rather than a deliberately formed hole. For this test, the number of faces around the void is compared to the smaller of a) three times the number of faces on one element, or b) one-tenth the total number of element faces in the model.

Menu Paths

Main Menu>Preprocessor>Meshing>Check Mesh>Connectivity>Ck Connectvty

Main Menu>Preprocessor>Meshing>Check Mesh>Connectivity>Sel Bad Connt

---

MCFOPT,*Format*,*Type*,*Norm*

Specifies options for the Modal Coordinates File (`Jobname.mcf`).

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Format

`Jobname.mcf` file format:

0

ASCII with wrap for more than 25 values (harmonic analysis) or 50 values (transient analysis) (default).

1

ASCII.

Type

Output form of the complex modal coordinates – Only supported for *Format* = 1:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Real and imaginary parts (default).

1

Amplitude and phase angle in degree.

Norm

Mode shape normalization for the calculation of the modal coordinates – Only supported for *Format* = 1:

0

The modes are mass normalized (default).

1

The modes are normalized to unity.

Notes

Options specified with **MCFOPT (p. 1116)** are processed when a request is made to write the *Jobname.mcf* file by issuing **TRNOPT (p. 2017)** or **HROPT (p. 912)** with *MCFwrite* = ON.

If you specify normalized to unity for the modal coordinates (*Norm* = 1), the generalized mass must be available (see Modal Mass and Kinetic Energy Output in the *Structural Analysis Guide*). Note that this option is independent from the normalization of the modes specified during the modal analysis (*NrmKey* on **MODOPT (p. 1140)** command).

Menu Paths

This command cannot be accessed from a menu.

MDAMP, *STLOC*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Defines the damping ratios as a function of mode.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

STLOC

Starting location in table for entering data. For example, if *STLOC* = 1, data input in the *V1* field applies to the first constant in the table. If *STLOC* = 7, data input in the *V1* field applies to the seventh constant in the table, etc. Defaults to the last location filled + 1.

V1, V2, V3, . . . , V6

Data assigned to six locations starting with *STLOC*. If a value is already in this location, it will be redefined. Blank values for *V2* to *V6* leave the corresponding previous value unchanged.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines the damping ratios as a function of mode. Table position corresponds to mode number. These ratios are added to the **DMPRAT (p. 562)** value, if defined. Use the **STAT (p. 1861)** command to list current values. This command applies to mode-superposition harmonic (**ANTYPE (p. 162)**,HARMIC), mode-superposition linear transient dynamic (**ANTYPE (p. 162)**,TRANS), and spectrum (**ANTYPE (p. 162)**,SPECTR) analyses. Repeat the **MDAMP (p. 1117)** command for additional constants (10000 maximum).

MDAMP (p. 1117) can also be defined in a substructure analysis that uses component mode synthesis. The damping ratios are added on the diagonal of the reduced damping matrix, as explained in Component Mode Synthesis (CMS).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Damping

Main Menu>Solution>Load Step Opts>Time/Frequenc>Damping

MDELE, *NODE*, *Lab1*, *NEND*, *NINC*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *SUPPORT*

Deletes master degrees of freedom.

SOLUTION (p. 38): Master DOF (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE*, *Lab1*, *NEND*, *NINC

Delete master degrees of freedom in the *Lab1* direction (**M (p. 1097)**) from *NODE* to *NEND* (defaults to *NODE*) in steps of *NINC* (defaults to 1). If *NODE* = ALL, *NEND* and *NINC* are ignored and masters for all selected nodes (**NSEL (p. 1266)**) are deleted. If *Lab1* = ALL, all label directions will be deleted. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

Lab2*, *Lab3*, *Lab4*, . . . , *Lab6

Delete masters in these additional directions.

SUPPORT

Pseudo-constraints key for the free-interface (**CMSOPT (p. 378)**,FREE) and residual-flexible free-interface (**CMSOPT (p. 378)**,RFFB) CMS method analyses:

OFF – delete selected master degrees of freedom and any pseudo-constraints applied on them with SUPPORT = ON in the **M (p. 1097)** command (default).

ON – only delete any pseudo-constraints applied on selected master degrees of freedom.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Deletes master degrees of freedom. If used in SOLUTION, this command is valid only within the first load step.

The *SUPPORT* argument is ignored for the fixed-interface CMS method analysis (**CMSOPT (p. 378)**,FIX).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>Delete

Main Menu>Solution>Master DOFs>Delete

MDPLOT,*Function*,*Dmpname*,*Scale*

Plots frequency-dependent modal damping coefficients calculated by DMPEXT.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Function

Function to display.

d_coeff

Damping coefficient

s_coeff

Squeeze coefficient

d_ratio

Damping ratio

s_ratio

Squeeze stiffness ratio

Dmpname

Array parameter name where damping information is stored. Defaults to d_damp.

Scale

Indicates whether to perform a linear or a double logarithmic plot.

LIN

Perform a linear plot. Default

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LOG

Perform a double logarithmic plot.

Notes

See Thin Film Analysis for more information on thin film analyses.

Menu Paths

Main Menu>General Postproc>Plot Results>ThinFilm>Plot Dmp Parm

---

MEMM, *Lab*, *Keyword*

Allows the current session to keep allocated memory

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

When *Lab* = KEEP, the memory manager's ability to acquire and keep memory is controlled by *Keyword*

Keyword

Turns the memory "keep" mode on or off

ON

Keep any memory allocated during the analysis.

OFF

Use memory dynamically and free it up to other users after use (default).

Command Default

Keyword = OFF.

Notes

You can use the **MEMM (p. 1120)** command to ensure that memory intensive operations will always have the same memory available when the operations occur intermittently. Normally, if a large amount of memory is allocated for a specific operation, it will be returned to the system once the operation is finished. This option always maintains the highest level used during the analysis until the analysis is finished.

The **MEMM (p. 1120)** command does not affect the value you specify with the `-m` switch. When you allocate memory with the `-m` switch, that amount will always be available. However, if dynamic memory

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

allocation in excess of the -m value occurs, you can use the **MEMM (p. 1120)** command to ensure that amount is retained until the end of your analysis.

Menu Paths

This command cannot be accessed from a menu.

/MENU, *Key*

Activates the Graphical User Interface (GUI).

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Activation key:

ON

Activates the menu system (device dependent).

Command Default

GUI is enabled if entering Mechanical APDL program via the launcher. GUI is disabled if entering via the Mechanical APDL execution command.

Notes

Activates the Graphical User Interface (GUI).

Caution:

if you include the **/MENU (p. 1121)**,ON command in your `start.ans`, it should be the *last* command in the file. Any commands after **/MENU (p. 1121)**,ON may be ignored. (It is not necessary to include the **/SHOW (p. 1785)** and **/MENU (p. 1121)**,ON commands in `start.ans` if you will be using the launcher to enter the Mechanical APDL program.)

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*MERGE, *Name1*, *Name2*, *Val1*, *Val2*

Merges two dense matrices or vectors into one.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name1

Name of the matrix or vector to extend.

Name2

Name of the matrix or vector to be merged into *Name1*.

Val1*, *Val2

Additional input. The meaning of *Val1* and *Val2* varies depending on the entity type (matrix or vector). See details below.

The following *Val1* and *Val2* fields are used if *Name1* refers to a dense matrix created by the *DMAT (p. 551) command:

Val1

Column or row number indicating where the new values are to be inserted into the *Name1* matrix.

Val2

Specifies how the *Name2* matrix or vector is copied into the *Name1* matrix.

COL

Insert the new values at the column location specified by *Val1* (default).

ROW

Insert the new values at the row location specified by *Val1*.

The following *Val1* field is used if *Name1* refers to a vector created by the *VEC (p. 2085) command:

Val1

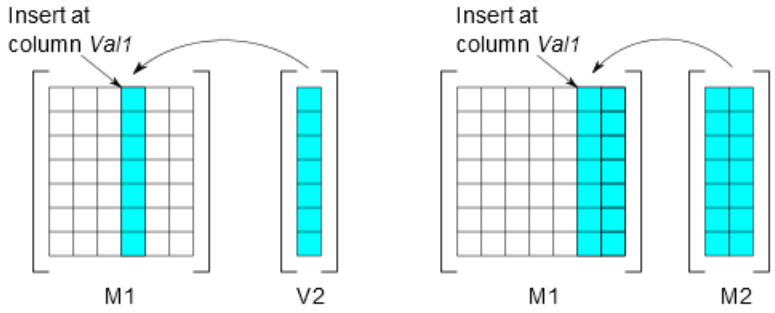
Row number indicating where the new values are to be inserted into the *Name1* vector.

Notes

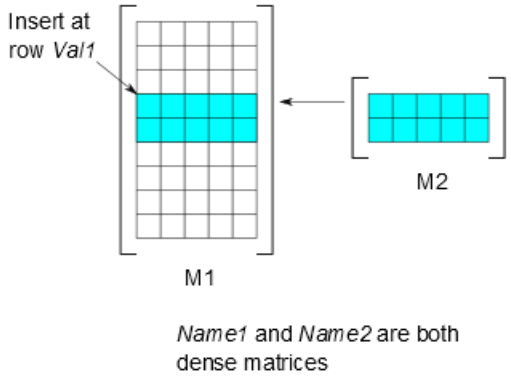
*MERGE (p. 1122) can be used to add new columns or rows to a dense matrix that was created by the *DMAT (p. 551) command. In this case, *Name1* must be the name of the dense matrix and *Name2* must refer to a vector or another dense matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

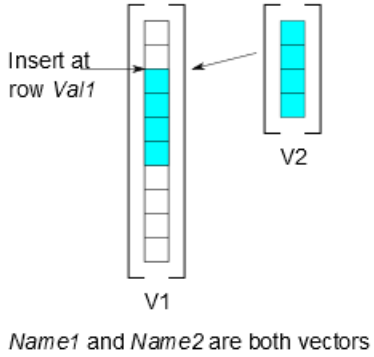
The following two examples demonstrate merging columns into a dense matrix.



The following example demonstrates merging rows into a dense matrix.



***MERGE (p. 1122)** can also be used to add new rows to a vector that was created by the ***VEC (p. 2085)** command. In this case, *Name1* and *Name2* must both refer to vectors, as demonstrated in the example below.



In all cases, the values of the original matrix or vector are retained, and the matrix or vector is resized to accommodate the additional rows or columns.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

MESHING

Specifies "Meshing" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Meshing

*MFOURI, *Oper*, *COEFF*, *MODE*, *ISYM*, *THETA*, *CURVE*

Calculates the coefficients for, or evaluates, a Fourier series.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Oper

Type of Fourier operation:

FIT --

Calculate Fourier coefficients *COEFF* from *MODE*, *ISYM*, *THETA*, and *CURVE*.

EVAL --

Evaluate the Fourier curve *CURVE* from *COEFF*, *MODE*, *ISYM* and *THETA*

COEFF

Name of the array parameter vector containing the Fourier coefficients (calculated if *Oper* = FIT, required as input if *Oper* = EVAL). See ***SET (p. 1720)** for name restrictions.

MODE

Name of the array parameter vector containing the mode numbers of the desired Fourier terms.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ISYM

Name of the array parameter vector containing the symmetry key for the corresponding Fourier terms. The vector should contain keys for each term as follows:

0 or 1 --

Symmetric (cosine) term

-1 --

Antisymmetric (sine) term.

THETA, CURVE

Names of the array parameter vectors containing the theta vs. curve description, respectively. Theta values should be input in degrees. If *Oper* = FIT, one curve value should be supplied with each theta value. If *Oper* = EVAL, one curve value will be calculated for each theta value.

Notes

Calculates the coefficients of a Fourier series for a given curve, or evaluates the Fourier curve from the given (or previously calculated) coefficients. The lengths of the *COEFF*, *MODE*, and *ISYM* vectors must be the same--typically two times the number of modes desired, since two terms (sine and cosine) are generally required for each mode. The lengths of the *CURVE* and *THETA* vectors should be the same or the smaller of the two will be used. There should be a sufficient number of points to adequately define the curve--at least two times the number of coefficients. A starting array element number (1) must be defined for each array parameter vector. The vector specifications ***VLEN (p. 2136)**, ***VCOL (p. 2077)**, ***VABS (p. 2072)**, ***VFACT (p. 2092)**, and ***VCUM (p. 2080)** do not apply to this command. Array elements should not be skipped with the ***VMASK (p. 2139)** and the *NINC* value of the ***VLEN (p. 2136)** specifications. The vector being calculated (*COEFF* if *Oper* is FIT, or *CURVE* if *Oper* is EVAL) must exist as a dimensioned array (***DIM (p. 530)**).

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Matrix Fourier

***MFUN**, *ParR*, *Func*, *Par1*

Copies or transposes an array parameter matrix.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting array parameter matrix. See ***SET (p. 1720)** for name restrictions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Func

Copy or transpose function:

COPY --

Par1 is copied to *ParR*

TRAN --

Par1 is transposed to *ParR*. Rows (m) and columns (n) of *Par1* matrix are transposed to resulting *ParR* matrix of shape (n,m).

Par1

Array parameter matrix input to the operation.

Notes

Operates on one input array parameter matrix and produces one output array parameter matrix according to:

ParR = f(*Par1*)

where the function (f) is either a copy or transpose, as described above.

Functions are based on the standard FORTRAN definitions where possible. *ParR* may be the same as *Par1*. Starting array element numbers must be defined for each array parameter matrix if it does not start at the first location. For example, ***MFUN (p. 1125)**,A(1,5),COPY,B(2,3) copies matrix B (starting at element (2,3)) to matrix A (starting at element (1,5)). The diagonal corner elements for each submatrix must be defined: the upper left corner by the array starting element (on this command), the lower right corner by the current values from the ***VCOL (p. 2077)** and ***VLEN (p. 2136)** commands. The default values are the (1,1) element and the last element in the matrix. No operations progress across matrix planes (in the 3rd dimension). Absolute values and scale factors may be applied to all parameters (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). Array elements should not be skipped with the ***VMASK (p. 2139)** and the *NINC* value of the ***VLEN (p. 2136)** specifications. The number of rows (***VLEN (p. 2136)**) applies to the *Par1* array. See the ***VOPER (p. 2142)** command for details.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Matrix Functions

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MGEN, *ITIME*, *INC*, *NODE1*, *NODE2*, *NINC*

Generates additional MDOF from a previously defined set.

SOLUTION (p. 38): Master DOF (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME*, *INC

Do this generation operation a total of *ITIME*s, incrementing all nodes in the set by *INC* each time after the first. *ITIME* must be > 1 for generation to occur. All previously defined master degree of freedom directions are included in the set. A component name may also be substituted for *ITIME*.

NODE1*, *NODE2*, *NINC

Generate master degrees of freedom from set beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and set is all selected nodes (**NSEL (p. 1266)**). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Generates additional master degrees of freedom from a previously defined set. If used in SOLUTION, this command is valid only within the first load step.

For the free-interface (**CMSOPT (p. 378)**, FREE) and residual-flexible free-interface (**CMSOPT (p. 378)**, RFFB) CMS method analyses, pseudo-constraints could have been applied on some master degrees of freedom of the previously defined set (SUPPORT = ON in the **M (p. 1097)** command). The master degrees of freedom generated from these with the **MGEN (p. 1127)** command are also defined with pseudo-constraints.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>Copy

Main Menu>Solution>Master DOFs>Copy

MIDTOL, *KEY*, *TOLERB*, *RESFQ*

Sets midstep residual criterion values for structural transient analyses.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Midstep residual criterion activation key.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON or 1

Activate midstep residual criterion in a structural transient analysis (default).

OFF or 0

Deactivate midstep residual criterion in a structural transient analysis.

STAT

List the current midstep residual criterion setting.

T0LERB

Midstep residual tolerance or reference value for *bisection*. Defaults to 100 times the *T0LER* setting of the **CNVTOL (p. 396)** command.

If *T0LERB* > 0, it is used as a tolerance about the typical force and/or moment to compare midstep residual force and/or moment for convergence.

If *T0LERB* < 0, it is used as a reference force value against which the midstep residual force is compared for convergence. The reference force value is used to compute a reference moment value for midstep residual moment comparison.

If midstep residual force and/or moment has not converged and **AUTOTS (p. 207)**,ON is used, then *T0LERB* is also used to predict time step size for bisection.

RESFQ

Key to use response frequency computation along with midstep residual criterion for automatic time stepping (**AUTOTS (p. 207)**,ON).

OFF or 0

Do not calculate response frequency and do not consider it in the automatic time stepping (default).

ON or 1

Calculate response frequency and consider it in the automatic time stepping.

Command Default

For transient structural analysis, the out-of-balance residual is not checked at the midstep.

Notes

When *T0LERB* is input as a tolerance value (*T0LERB* > 0), the typical force and/or moment from the regular time step is used in the midstep residual force and/or moment comparison.

In a structural transient analysis, the suggested tolerance range of *T0LERB* (*T0LERB* > 0) is as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TOLERB = 1 to 10 times the *TOLER* setting of the **CNVTOL (p. 396)** command for high accuracy solution.

TOLERB = 10 to 100 times the *TOLER* setting of the **CNVTOL (p. 396)** command for medium accuracy solution.

TOLERB = more than 100 times the *TOLER* setting of the **CNVTOL (p. 396)** command for low accuracy solution.

If the structural transient analysis is elastic and linear, and the load is constant or changes slowly, use a smaller value of *TOLERB* to achieve an accurate solution. If the analysis involves large amounts of energy dissipation, such as elastic-plastic material, *TOLERB* can be larger. If the analysis includes contact or rapidly varying loads, a smaller value of *TOLERB* should be used if high frequency response is important; otherwise, a larger value of *TOLERB* may be used to enable faster convergence with larger time step sizes.

For more information on how the midstep criterion is used by the program, see Midstep Residual for Structural Dynamic Analysis in the *Mechanical APDL Theory Reference*.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient

Main Menu>Solution>Analysis Type>Sol'n Controls>Transient

/MKDIR, *Dir*

Creates a directory.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Dir

The directory to create (248 characters maximum on Linux; 233 on Windows). If no path is provided, it will be created in the current working directory. Must be a valid name (and path) for the system you are working on.

Notes

Creates a directory on the computer Mechanical APDL is currently running on.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MLIST,*NODE1*,*NODE2*,*NINC*

Lists the MDOF of freedom.

SOLUTION (p. 38): Master DOF (p. 46)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2*, *NINC

List master degrees of freedom from *NODE1* to *NODE2* (defaults to*NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and masters for all selected nodes (**NSEL (p. 1266)**) are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Lists the master degrees of freedom.

For the free-interface CMS method analysis (**CMSOPT (p. 378)**,FREE), any pseudo-constraints applied on master degrees of freedom with *SUPPORT* = ON in the **M (p. 1097)** command will be listed when **MLIST (p. 1130)** is issued after **CMSOPT (p. 378)** (see example printout below).

```
 NODE   LABEL      SUPPORT
  8529   UX
  8529   UY
  8529   UZ
  8545   UX          ON
  8545   UY          ON
  8545   UZ          ON
```

Menu Paths

Main Menu>Preprocessor>Loads>Master DOFs>List All

Main Menu>Preprocessor>Loads>Master DOFs>List Picked

Main Menu>Solution>Master DOFs>List All

Main Menu>Solution>Master DOFs>List Picked

Utility Menu>List>Other>Master DOF>At All Nodes

Utility Menu>List>Other>Master DOF>At Picked Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MMASS,*Option*,*ZPA*

Specifies the missing mass response calculation.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Flag to activate or deactivate missing mass response calculation.

0 (OFF or NO)

Deactivate (default).

1 (ON or YES)

Activate.

ZPA

Zero Period Acceleration Value. If a scale factor *FACT* is defined on the **SVTYP (p. 1887)** command, it is applied to this value.

Notes

The missing mass calculation is valid only for single point excitation response spectrum analysis (**SPOPT (p. 1837)**, SPRS) and for multiple point response spectrum analysis (**SPOPT (p. 1837)**, MPRS) performed with base excitation using acceleration response spectrum loading. Missing mass is supported in a spectrum analysis only when the preceding modal analysis is performed with the Block Lanczos, PCG Lanczos, Supernode, or Subspace eigensolver (Method =LANB, LANPCG, SNODE, or SUBSP on the **MODOPT (p. 1140)** command).

The velocity solution is not available (*Label* = VELO on the combination command: **SRSS (p. 1845)**, **CQC (p. 433)**...) when the missing mass calculation is activated.

The missing mass calculation is not supported when the spectrum analysis is based on a linear perturbation modal analysis performed after a nonlinear base analysis.

The missing mass is not supported when superelements are present.

To take into account the contribution of the truncated modes, the residual vector (**RESVEC (p. 1603)**) can be used in place of the missing mass response. This is of particular interest if the velocity solution is requested or if a nonlinear prestress is included in the analysis (linear perturbation), or if a superelement is present, since the missing mass cannot be used in these cases.

In a multiple point response spectrum analysis (**SPOPT (p. 1837)**,MPRS), the **MMASS (p. 1131)** command must precede the participation factor calculation command (**PFACT (p. 1385)**).

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional Information

Performing a Single-Point Response Spectrum (SPRS) Analysis in the *Structural Analysis Guide*

Performing a Multi-Point Response Spectrum (MPRS) Analysis in the *Structural Analysis Guide*

Missing-Mass Response in the *Mechanical APDL Theory Reference*

RIGRESP (p. 1610) command

Menu Paths

This command cannot be accessed from a menu.

MMF

Calculates the magnetomotive force along a path.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

MMF (p. 1132) invokes a Mechanical APDL macro which calculates the magnetomotive force (mmf) along a predefined path (**PATH (p. 1354)**). It is valid for both 2D and 3D magnetic field analyses. The calculated mmf value is stored in the parameter MMF.

A closed path (**PATH (p. 1354)**), passing through the magnetic circuit for which mmf is to be calculated, must be defined before this command is issued. A counterclockwise ordering of points on the **PPATH (p. 1463)** command will yield the correct sign on the mmf. The mmf is based on Ampere's Law. The macro makes use of calculated values of field intensity (H), and uses path operations for the calculations. All path items are cleared upon completion. The **MMF (p. 1132)** macro sets the "ACCURATE" mapping method and "MAT" discontinuity option of the **PMAP (p. 1449)** command.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Path Based>MMF

MODCONT,*MLSkey*,*EnforcedKey*,--,*FastLV*

Specify additional modal analysis options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MLSkey

Multiple load step key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Perform the modal analysis (compute the eigenvalues and the load vector) for each load step. (default)

ON

Perform the modal analysis (compute the eigenvalues and the load vector) only for the first load step; form the load vector for each subsequent load step (without repeating the eigenvalue calculations) and write all load vectors to the `Jobname.mode` file for downstream mode-superposition analyses.

EnforcedKey

Enforced motion key:

OFF

Do not calculate enforced static modes. (default)

ON

Calculate enforced static modes and write them to the `Jobname.mode` file.

--

Unused field

FastLV

Fast load vector generation key; valid only when *MLSkey* = ON:

OFF

Do not activate fast load vector generation (default).

ON

Activate fast load vector generation. This option is only supported when each load vector is based on a unique element surface load (**SFE (p. 1761)**) applied on one element, and the element result superposition key is activated (*MSUPkey* = YES on **MXPAND (p. 1203)**).

Notes

Specifies additional modal analysis (**ANTYPE (p. 162)**,MODAL) options.

Use the **LVSCALE (p. 1094)** command to apply the desired load in a mode-superposition transient or harmonic analysis.

When *MSUPkey* = YES on the **MXPAND (p. 1203)** command, the maximum number of load vectors allowed in the `Jobname.mode` file defaults to 1000. To increase this limit, use the command **/CONFIG (p. 410)**,NUMLV. When *FastLV* = ON, the limit is automatically set to $1x10^6$ and cannot be changed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The maximum number of load vectors that can be used in the downstream mode-superposition transient or harmonic analysis is the number of load vectors written in the Jobname.mode file.

Generation of multiple loads (*MLSkey* = ON) is supported by the Block Lanczos, PCG Lanczos, Supernode, Subspace, Unsymmetric, and QRDAMP mode extraction methods.

The enforced motion calculation (*EnforcedKey* = ON) is supported by the Block Lanczos, Supernode, Subspace, and QRDAMP mode extraction methods.

Menu Paths

This command cannot be accessed from a menu.

MODDIR, *Key*, *Directory*, *Fname*

Enables remote read-only usage of modal analysis files or substructuring analysis files.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Key for enabling remote read-only usage of modal analysis files or substructuring analysis files:

1 (ON or YES)

The program performs the analysis using remote files. The files are read-only.

0 (OFF or NO)

The program performs the analysis using files located in the working directory (default).

LIST

List remote modal files status, directory path, and file name.

Directory

Directory path (248 characters maximum). The directory contains the modal analysis files or the substructuring generation pass files.

The directory path defaults to the current working directory.

Fname

File name (no extension or directory path) for the modal analysis files or the substructuring generation pass files.

The file name defaults to the current Jobname.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This commands is used when solving linked analyses in two different folders. It is mostly meant to be used to reduce solution time and disk space usage by specifying the path to required solution files rather than copying them in the Mechanical Application (see the file management sections of the following analyses: mode-superposition transient, mode-superposition harmonic, response spectrum,random vibration, and substructure generation in the *Mechanical User's Guide*).

This command applies to the following analysis types:

- Spectrum (**ANTYPE (p. 162)**,SPECTR)
- Modal restart (**ANTYPE (p. 162)**, MODAL, RESTART)
- Mode-superposition transient (**ANTYPE (p. 162)**,TRANS and **TRNOPT (p. 2017)**, MSUP)
- Mode-superposition harmonic (**ANTYPE (p. 162)**,HARM and **HROPT (p. 912)**, MSUP)
- Substructuring (**ANTYPE (p. 162)**,SUBSTR).

Using the default for both the directory path (*Directory*) and the file name (*Fname*) is not valid. At least one of these values must be specified.

In a spectrum analysis and in mode-superposition analyses, **MODDIR (p. 1134)** must be issued during the first solution and at the beginning of the solution phase (before **LVSCALE (p. 1094)** in particular). In a spectrum analysis, remote modal files usage is not supported when mode file reuse is activated (*modeReuseKey* = YES on **SPOPT (p. 1837)**).

After a PSD spectrum analysis, **MODDIR (p. 1134)** can be issued in POST26 prior to the **STORE (p. 1867)** command.

In a modal restart analysis, **MODDIR (p. 1134)** must be issued during the first solution. Remote modal files usage cannot be activated in a modal restart analysis if during the first modal analysis:

- Enforced static modes have been calculated (*EnforcedKey* = ON on **MODCONT (p. 1132)**).
- Element result superposition key (*MSUPkey* on **MXPAND (p. 1203)**) was set to NO, whereas it is set to YES in the modal restart.

When using distributed-memory parallel processing, if element results calculation based on element modal results is activated for the spectrum analysis (*Elcalc* = YES on **SPOPT (p. 1837)**), **MODDIR (p. 1134)** usage can significantly reduce computation time as it decreases the size of the distributed `.rst` files to be combined at the end of spectrum analysis solution.

In a substructuring or CMS analysis, **MODDIR (p. 1134)** can be issued during either of the following analysis phases:

- The first solution of the first restart of a generation pass. *ExpMth* on **SEOPT (p. 1715)** must be set to MODDIR during the first solve of the primary generation pass.
- The first solution of the expansion pass.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

MODE, *MODE*, *ISYM*

Specifies the harmonic loading term for this load step.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MODE

Number of harmonic waves around circumference for this harmonic loading term (defaults to 0).

ISYM

Symmetry condition for this harmonic loading term (not used when *MODE* = 0):

1

Symmetric (UX, UY, ROTZ, TEMP use cosine terms; UZ uses sine term) (default).

-1

Antisymmetric (UX, UY, ROTZ, TEMP use sine terms; UZ uses cosine term).

Command Default

MODE = 0, *ISYM* = 1.

Notes

Used with axisymmetric elements having nonaxisymmetric loading capability (for example, PLANE25, SHELL61, etc.). For analysis types **ANTYPE (p. 162)**,MODAL, HARMIC, TRANS, and SUBSTR, the term must be defined in the first load step and may not be changed in succeeding load steps.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>For Harmonic Ele

Main Menu>Solution>Load Step Opts>Other>For Harmonic Ele

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MODIFY,*SET*,*LSTEP*,*ITER*,*CUMIT*,*TIME*,*Ktitle*

Changes the listed values of the data in a set.

AUX3 (p. 61): Results Files (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SET

Set of data in results file to be modified.

LSTEP

The new load step number.

ITER

The new load substep number.

CUMIT

The new cumulative iteration.

TIME

The new time/frequency value.

Ktitle

Indicates if the set title should be modified.

0

Keep the original title.

1

Change the title to the title specified with the most current **/TITLE (p. 2007)** command.

Notes

Use this command to change the listed values in a data set in a results file. Using this command does not change any actual model data; it affects only the values listed in the results file.

The **MODIFY (p. 1137)** command is valid only in the results file editing processor (auxiliary processor AUX3), and, like the other AUX3 commands, it only affects the data steps index (DSI), time (TIM), loadstep, substep and cumulative step iteration (LSP) records in the results file.

Example Usage

If you start with the following results file:

```
SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
 1     1.0000          1          1         1
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 first load set

 2     2.0000           2          1         2
 second load set

 3     3.0000           3          1         3
 third load set

 4     4.0000           4          1         4
 fourth load set
```

and you then issue the following commands:

```
/title, modified title for set number 3
modify,3,5,2,5,4.5,1
```

The modified results file would look like this:

```
SET   TIME/FREQ    LOAD STEP   SUBSTEP  CUMULATIVE
 1     1.0000          1          1         1
 first load set

 2     2.0000           2          1         2
 second load set

 3     4.5000           5          2         5
 modified title for set number 3

 4     4.0000           4          1         4
 fourth load set
```

Menu Paths

Main Menu>Preprocessor>Loads>Other>For Harmonic Ele

Main Menu>Solution>Other>For Harmonic Ele

MODMSH, *Lab*

Controls the relationship of the solid model and the FE model.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Relationship key:

STAT

Gives status of command (default). This applies only to the CHECK option (no status is provided for the DETACH option).

NOCHECK

Deactivates the checking of the solid model and the finite element model. Allows elements and nodes generated with the mesh commands to be modified directly (**EMODIF (p. 626)**,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NMODIF (p. 1247), **EDELE (p. 593)**, **NDELE (p. 1211)**, etc.). Also deactivates solid model hierarchical checking so that areas attached to volumes may be deleted etc.

CHECK

Reactivates future checking of the solid model.

DETACH

Releases all associativity between the current solid model and finite element model. Mechanical APDL deletes any element attributes that were assigned to the affected solid model entities through default attributes (that is, through the **TYPE (p. 2036)**, **REAL (p. 1579)**, **MAT (p. 1111)**, **SECNUM (p. 1685)**, and **ESYS (p. 685)** command settings and a subsequent meshing operation). However, attributes that were assigned directly to the solid model entities (via the **KATT (p. 969)**, **LATT (p. 1014)**, **AATT (p. 85)**, and **VATT (p. 2075)** commands) are not deleted.

Notes

Affects the relationship of the solid model (keypoints, lines, areas, volumes) and the finite element model (nodes, elements, and boundary conditions).

Specify *Lab* = NOCHECK carefully. By deactivating checking, the solid model database can be corrupted by subsequent operations.

After specifying *Lab* = DETACH, it is no longer possible to select or define finite element model items in terms of the detached solid model or to clear the mesh.

Menu Paths

Main Menu>Preprocessor>Checking Ctrls>Model Checking

MODSELOPTION, *dir1*, *dir2*, *dir3*, *dir4*, *dir5*, *dir6*

Specifies the criteria for selecting the modes to be expanded.

SOLUTION (p. 38): Dynamic Options (p. 40)
SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

dir1, dir2, …, dir6

Selection of the direction to be expanded.

For *ModeSelMethod* = EFFM on the **MXPAND (p. 1203)** command, the directions correspond to the global Cartesian directions, i.e. 1=X, 2=Y, 3=Z, 4=ROTX, 5=ROTY, and 6=ROTZ. If *dir1* = YES, then any mode in this direction is expanded if its modal effective mass divided by the total mass (modal effective mass ratio) is greater than *SIGNIF* on the **MXPAND (p. 1203)** command. If *dir1*=NO, then the specified direction is not considered as a criterion for expansion. If *dir1* is given a numerical decimal value, modes in that direction are selected (starting from the ones with the largest modal

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

effective mass ratios to the smallest) until the sum of their modal effective mass ratio equals this requested threshold.

For *ModeSelMethod* = MODC on the **MXPAND (p. 1203)** command, *dir1* corresponds to the first input spectrum, *dir2* to the second, etc. (i.e. for multiple spectrum inputs; the actual directions correspond to their respective SED directions). If *dir1*=YES, then any mode in this spectrum is expanded if its mode coefficient divided by the largest mode coefficient is greater than SIGNIF on the **MXPAND (p. 1203)** command. If *dir1*=NO, then the specified direction is not considered as a criterion for expansion.

Notes

This command is only applicable when a mode selection method is defined (*ModeSelMethod* on the **MXPAND (p. 1203)** command). See Using Mode Selection in the *Structural Analysis Guide* for more details.

If a numerical value is specified for a direction, the significance threshold (SIGNIF on the **MXPAND (p. 1203)** command) is ignored for the selection of the modes in this direction.

If a mode is determined to be expanded in **any** of the 6 directions, it will be expanded in the `.mode` file. Otherwise, the mode will not be expanded.

The default behavior is to consider all directions for expansion.

For *ModeSelMethod* = EFFM on the **MXPAND (p. 1203)** command, *dir4*, *dir5*, and *dir6* are not considered if mass related information have been calculated using the lumped approximation instead of the precise calculation (See Mass Related Information in the *Theory Reference*).

Menu Paths

This command cannot be accessed from a menu.

MODOPT,*Method*,*NMODE*,*FREQB*,*FREQE*,*Cpxmod*,*Nrmkey*,*ModType*,*BlockSize*,--,--,--,*FREQMOD*

Specifies modal analysis options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1144)

Method

Mode-extraction method to be used for the modal analysis.

LANB

Block Lanczos

LANPCG

PCG Lanczos

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SNODE

Supernode modal solver

SUBSP

Subspace algorithm

UNSYM

Unsymmetric matrix

DAMP

Damped system

QRDAMP

Damped system using QR algorithm

NMODE

The number of modes to extract. The value can depend on the value supplied for *Method*. *NMODE* has no default and must be specified. If *Method* = LANB, LANPCG, or SNODE, the number of modes that can be extracted can equal the DOFs in the model after the application of all boundary conditions.

For *Method* = LANB, SUBSP and UNSYM, you can specify *NMODE* = ALL to extract all eigenvalues in a given frequency range. To use this option, you must also define the frequency range (that is, a *FREQE* value must be set).

Recommendation:

When *Method* = LANPCG, *NMODE* should be less than 100 to be computationally efficient.

When *Method* = SNODE, *NMODE* should be greater than 100 for 2D plane or 3D shell/beam models and greater than 250 for 3D solid elements to be computationally efficient.

FREQB

The beginning, or lower end, of the frequency range (or eigenvalue range if *FREQMOD* is specified) of interest.

For *Method* = LANB, SUBSP, UNSYM, DAMP, and QRDAMP, *FREQB* also represents the first shift point for the eigenvalue iterations. For UNSYM and DAMP, the default = -1.0 For other methods, the default is calculated internally.

Eigenvalue extraction is most accurate near the shift point. Multiple shift points are used internally in the LANB, SUBSP, UNSYM, and QRDAMP methods. For the LANB, LANPCG, SUBSP, UNSYM, DAMP, and QRDAMP methods with a positive *FREQB* value, eigenvalues are output beginning at the shift point and increase in magnitude. For the UNSYM and DAMP methods with a negative *FREQB* value, eigenvalues are output beginning at zero magnitude and increase.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Choosing higher *FREQB* values with the LANPCG and SNODE methods may lead to inefficient solution times because these methods will find all eigenvalues between zero and *FREQB* before finding the requested modes between *FREQB* and *FREQE*.

FREQE

The ending, or upper end, of the frequency range (or eigenvalue range if *FREQMOD* is specified) of interest (in Hz). Default = 100 Hz when *Method* = SNODE. The default for all other methods is to calculate all modes, regardless of their maximum frequency.

To maintain solution efficiency, do not set the *FREQE* value too high; for example, not higher than 5000 Hz for an industrial problem. The higher the *FREQE* value used for the SNODE method, the more accurate the solution and the more eigenvalues produced; however, the solution time also increases. For example, if *FREQE* is set to 1e8, it causes the underlying supernodal structures to find all possible eigenvalues of each group of supernodes, requiring excessive solution time. The accuracy of the SNODE solution is controlled by *FREQE* and by the *RangeFact* value on the **SNOPTION (p. 1815)** command.

Cpxmod

Complex eigenmode key. (Valid only when *Method* = QRDAMP or *Method* = UNSYM).

AUTO

Determine automatically if the eigensolutions are real or complex and output them accordingly (default when *Method* = UNSYM). Not supported for *Method* = QRDAMP.

ON or CPLX

Calculate and output complex eigenmode shapes.

OFF or REAL

Do not calculate complex eigenmode shapes (default). This setting is required if a mode-superposition analysis is intended after the modal analysis for *Method* = QRDAMP.

Nrmkey

Mode shape normalization key:

OFF

Normalize the mode shapes to the mass matrix (default). This option is invalid for damped modal cyclic symmetry (*Method* = DAMP or QRDAMP with the **CYCLIC (p. 462)** command).

ON

Normalize the mode shapes to unity instead of to the mass matrix (default for damped modal cyclic symmetry [*Method* = DAMP or QRDAMP with the **CYCLIC (p. 462)** command]).

If a subsequent spectrum or mode-superposition analysis is planned, the mode shapes should be normalized to the mass matrix (*Nrmkey* = OFF).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ModType

Type of modes calculated by the eigensolver. Only applicable to the unsymmetric eigensolver.

Blank

Right eigenmodes (default).

BOTH

Right and left eigenmodes. The left eigenmodes are written to `Jobname.lmode`. This option must be activated if a mode-superposition analysis is intended.

BlockSize

The block vector size to be used with the Block Lanczos or Subspace eigensolver (used only when *Method* = LANB or SUBSP). *BlockSize* must be an integer value between 0 and 16. When BlockSize = zero or blank, the code decides the block size internally (normally, a value of 8 is used for LANB and a value of 6 is used for SUBSP). Typically, higher *BlockSize* values are more efficient under each of the following conditions:

- When running in out-of-core mode and there is not enough physical memory to buffer all of the files written by the Block Lanczos or Subspace eigensolver (and thus, the time spent doing I/O is considerable).
- Many modes are requested (>100).
- Higher-order solid elements dominate the model.

The memory usage only slightly increases as *BlockSize* is increased. It is recommended that you use a value divisible by 4 (4, 8, 12, or 16).

--

Unused field.

--

Unused field.

--

Unused field.

FREQMOD

The specified frequency when the solved eigenvalues are no longer frequencies (for example, the model has the Floquet periodic boundary condition). In a modal analysis, the Floquet periodic boundary condition (body load FPBC) is only valid for the acoustic elements FLUID30, FLUID220, and FLUID221.

Notes

Specifies modal analysis (**ANTYPE (p. 162)**,MODAL) options.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional options for specific eigensolvers are controlled by these commands:

- **SNOPTION (p. 1815)** specifies options for the Supernode (SNODE) eigensolver.
- **SUBOPT (p. 1869)** specifies options for the Subspace (SUBSP) eigensolver.
- **QRDOPT (p. 1560)** specifies options for the QRDAMP eigensolver.
- **DAMPOPT (p. 495)** specifies options for the damped (DAMP) eigensolver.
- **LANBOPTION (p. 1007)** specifies options for the Block Lanczos (LANB) eigensolver. For more difficult modal problems, you can specify an alternative version of the Block Lanczos eigensolver (**LANBOPTION (p. 1007)**,,, ALT1).

If *Method* = LANPCG, Mechanical APDL automatically switches to the PCG solver internally for this modal analysis. You can further control the efficiency of the PCG solver with the **PCGOPT (p. 1365)** and **EQSLV (p. 648)** commands.

For models that involve a non-symmetric element stiffness matrix, as in the case of a contact element with frictional contact, the QRDAMP eigensolver (**MODOPT (p. 1140)**, QRDAMP) extracts modes in the modal subspace formed by the eigenmodes from the symmetrized eigenproblem. The QRDAMP eigensolver symmetrizes the element stiffness matrix on the first pass of the eigensolution, and in the second pass, eigenmodes are extracted in the modal subspace of the first eigensolution pass. For such non-symmetric eigenproblems, you should verify the eigenvalue and eigenmode results using the non-symmetric matrix eigensolver (**MODOPT (p. 1140)**,UNSYM).

The DAMP and QRDAMP options cannot be followed by a subsequent spectrum analysis. The UNSYM method supports spectrum analysis when eigensolutions are real.

In a modal analysis, the Floquet periodic boundary condition (body load FPBC) is only valid for the acoustic elements FLUID30, FLUID220, and FLUID221.

For more details about mode shape normalization, see Description of Analysis for Symmetric Undamped Systems in the *Theory Reference*.

When rigid bodies are present in a modal analysis using the supernode modal eigensolver (**ANTYPE (p. 162)**,MODAL and **MODOPT (p. 1140)**,SNODE), mode shape displacements at the constrained nodes cannot be post-processed.

This command is also valid in PREP7.

Distributed-Memory Parallel (DMP) Restriction

All extraction methods are supported in a DMP solution. Block Lanczos, PCG Lanczos, SUBSP, UNSYM, DAMP, and QRDAMP are distributed eigensolvers that run a fully distributed solution. However, the Supernode eigensolver is not a distributed eigensolver; therefore, you will not see the full performance improvements with this method that you would with a fully distributed solution.

Product Restrictions

Command Option Method	Available Products

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LANB	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
LANPCG	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SNODE	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SUBSP	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
UNSYM	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DAMP	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
QRDAMP	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

MONITOR, *VAR*, *Node*, *Lab*

Controls contents of variable fields in the nonlinear solution monitor file.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAR

One of four variable field numbers in the monitor file whose contents can be specified by the Lab field. Valid arguments are integers 1, 2, 3, or 4. See Notes section for default values.

Node

The node number for which information is monitored in the specified *VAR* field. In the GUI, if *Node* = P, graphical picking is enabled. If blank, the monitor file lists the maximum value of the specified quantity (Lab field) for the entire structure.

Lab

The solution quantity to be monitored in the specified *VAR* field. Valid labels for solution quantities are UX, UY, and UZ (displacements); ROTX, ROTY, and ROTZ (rotations); and TEMP (temperature). Valid labels for reaction force are FX, FY, and FZ (structural force) and MX, MY, and MZ (structural moment). Valid label for heat flow rate is HEAT. For defaults see the Notes section.

Notes

The monitor file always has an extension of **.mntr**, and takes its file name from the specified `Jobname`. If no `Jobname` is specified, the file name defaults to `file`.

You must issue this command once for each solution quantity you want to monitor at a specified node at each load step. You cannot monitor a reaction force during a linear analysis. The variable field contents can be redefined at each load step by reissuing the command. The monitored quantities are appended to the file for each load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction forces reported in the monitor file may be incorrect if the degree of freedom of the specified node is involved in externally defined coupling (**CP (p. 421)** command) or constraint equations (**CE (p. 297)** command), or if the program has applied constraint equations internally to the node.

The following example shows the format of a monitor file. Note that the file only records the solution substep history when a substep is convergent.

```
SOLUTION HISTORY INFORMATION FOR JOB: file.mntr


LOAD   SUB-  NO.  NO.    TOTL  INCREMENT    TOTAL         VARIAB 1     VARIAB 2     VARIAB 3     VARIAB 4
  STEP   STEP ATTMP ITER   ITER  TIME/LFACT   TIME/LFACT    MONITOR      MONITOR      MONITOR      MONITOR
                                                            Wall         MxDs         FY           MxRe

     1      1    1     5      5    0.36000E-01  0.36000E-01   0.0000     -0.47242E-01 -0.13783E-01  0.22670E-04
     1      2    1     3      8    0.36000E-01  0.72000E-01   0.0000     -0.91552E-01 -0.27269E-01  0.38849E-03
     1      3    1     4     12    0.54000E-01  0.12600       0.0000     -0.15161     -0.43972E-01  0.25235E-03
     1      4    1     3     15    0.81000E-01  0.20700       0.0000     -0.22926     -0.65119E-01  0.11131E-03
     1      5    1     5     20    0.12150      0.32850       0.0000     -0.34188     -0.93242E-01  0.12267E-02
     1      6    1     4     24    0.12150      0.45000       0.0000     -0.46860     -0.11992      0.22689E-02
     1      7    1     5     29    0.18000      0.63000       0.0000     -0.65720     -0.16217      0.12111E-03
     1      8    2     7     38    0.63000E-01  0.69300       0.0000     -0.72594     -0.46582      0.67121E-02
     1      9    1    11     49    0.63000E-01  0.75600       0.0000     -0.79976      -1.1070      0.29302E-02
     1     10    1    10     59    0.63000E-01  0.81900       0.0000     -0.87073      -1.8708      0.95828E-02
     1     11    1    18     77    0.81000E-01  0.90000       0.0000     -0.90000      -269.31      0.73911
```

The following details the contents of the various fields in the monitor file:

LOAD STEP

The current load step number.

SUBSTEP

The current substep (time step) number.

NO. ATTEMPT

The number of attempts made in solving the current substep. This number is equal to the number of failed attempts (bisections) plus one (the successful attempt).

NO. ITER

The number of iterations used by the last successful attempt.

TOTL. ITER

Total cumulative number of iterations (including each iteration used by a bisection).

INCREMENT

TIME/LFACT

Time or load factor increments for the current substep.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TOTAL TIME/LFACT

Total time (or load factor) for the last successful attempt in the current substep.

VARIAB 1

Variable field 1. By default, this field lists the elapsed (or wall clock) times used up to (but not including) the current substep.

VARIAB 2

Variable field 2. In this example, the field is reporting the MZ value. By default, this field lists the maximum displacement in the entire structure.

VARIAB 3

Variable field 3. In this example, the field is reporting the FY value of a certain node. By default, this field reports the maximum equivalent plastic strain increment in the entire structure.

VARIAB 4

Variable field 4. By default, this field reports the maximum residual force in the entire structure.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Monitor

Main Menu>Solution>Load Step Opts>Nonlinear>Monitor

---

*MOPER, *ParR*, *Par1*, *Oper*, *Val1*, *Val2*, *Val3*, *Val4*, *Val5*, *Val6*

Performs matrix operations on array parameter matrices.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting array parameter matrix. See ***SET (p. 1720)** for name restrictions.

Par1

First array parameter matrix input to the operation.

Oper

Matrix operations. Usage of the *Val1* through *Val6* arguments varies for each operation, as described below:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INVERT --

(***MOPER (p. 1147)**, *ParR*, *Par1*, INVERT)

Square matrix invert: Inverts the *n* x *n* matrix in *Par1* into *ParR*. The matrix must be well conditioned.

> **Warning:**
>
> Non-independent or ill-conditioned equations can cause erroneous results.

For large matrices, use the APDL Math operation ***LSFACTOR (p. 1078)** for efficiency (see APDL Math).

MULT --

(***MOPER (p. 1147)**, *ParR*, *Par1*, MULT, *Par2*)

Matrix multiply: Multiplies *Par1* by *Par2*. The number of rows of *Par2* must equal the number of columns of *Par1* for the operation. If *Par2* is input with a number of rows greater than the number of columns of *Par1*, matrices are still multiplied. However, the operation only uses a number of rows of *Par2* equal to the number of columns of *Par1*.

COVAR --

(***MOPER (p. 1147)**, *ParR*, *Par1*, COVAR, *Par2*)

Covariance: The measure of association between columns of the *Par1* input matrix. *Par1* of size *m* runs (rows) by *n* data (columns) is first processed to produce a row vector containing the mean of each column, which is transposed to the output column vector *Par2* of *n* array elements. The *Par1* and *Par2* operation then produces *ParR*, a resulting *n* x *n* matrix of covariances (with the variances as the diagonal terms).

CORR --

(***MOPER (p. 1147)**, *ParR*, *Par1*, CORR, *Par2*)

Correlation: The correlation coefficient between columns of the *Par1* input matrix. *Par1* of size *m* runs (rows) by *n* data (columns) is first processed to produce a row vector containing the mean of each column, which is then transposed to the output column vector *Par2* of *n* array elements. The *Par1* and *Par2* operation then produces *ParR*, a resulting *n* x *n* matrix of correlation coefficients (with a value of 1.0 for the diagonal terms).

SOLV --

(***MOPER (p. 1147)**, *ParR*, *Par1*, SOLV, *Par2*)

Solution of simultaneous equations: Solves the set of *n* equations of *n* terms of the form $a_{n1}x_1 + a_{n2}x_2 + \cdots + a_{nn}x_n = b_n$ where *Par1* contains the matrix of a-coefficients, *Par2*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

contains the vector(s) of b-values, and *ParR* contains the vector(s) of x-results. *Par1* must be a square matrix. The equations must be linear, independent, and well conditioned.

> **Warning:**
>
> Non-independent or ill-conditioned equations can cause erroneous results.

For large matrices, use the APDL Math operation ***LSFACTOR (p. 1078)** for efficiency (see APDL Math).

SORT --

(***MOPER (p. 1147)**, *ParR*, *Par1*, SORT, *Par2*, *n1*, *n2*, *n3*)

Matrix sort: Sorts the columns of matrix *Par1* according to sort vector *Par2* and places the result back into *Par1*. Rows of *Par1* are moved to the corresponding positions indicated by the values of *Par2*. *Par2* may be a column of *Par1* (in which case it will also be re-ordered). Alternatively, you may specify the column of *Par1* to sort using *n1* (leaving *Par2* blank). A secondary sort can be specified by column *n2*, and a third sort using column *n3*. *ParR* is the vector of initial row positions (the permutation vector). Sorting *Par1* according to *ParR* should reproduce the initial ordering.

NNEAR --

(***MOPER (p. 1147)**, *ParR*, *Par1*, NNEAR, *Toler*)

Nearest Node: Finds the nodes nearest to the given set of points in *Par1*, where *Par1* is an *n* x 3 array of coordinate locations. *ParR* is a vector of the nearest selected nodes, or 0 if no nodes are nearer than *Toler*. *Toler* defaults to 1 and is limited to the maximum model size.

ENEAR --

(***MOPER (p. 1147)**, *ParR*, *Par1*, ENEAR, *Toler*)

Nearest Element: Finds the elements nearest to the given set of points in *Par1*, where *Par1* is an *n* x 3 array of coordinate locations. *ParR* is a vector of the nearest selected elements, or 0 if no element centroids are nearer than *Toler*. *Toler* defaults to 1 and is limited to the maximum model size.

MAP --

(***MOPER (p. 1147)**, *ParR*, *Par1*, MAP, *Par2*, *Par3*, *kDim*, *--*, *kOut*, *LIMIT*)

Maps the results from one set of points to another. For example, you can map pressures from a CFD analysis onto your model for a structural analysis.

Par1 is the *Nout* x 3 array of points that will be mapped to. *Par2* is the *Nin* x *M* array that contains *M* values of data to be interpolated at each point and corresponds to the *Nin* x 3 points in *Par3*. The resulting *ParR* is the *Nout* x *M* array of mapped data points.

For each point in the destination mesh, all possible triangles in the source mesh are searched to find the best triangle containing each point. It then does a linear interpolation inside

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

this triangle. You should carefully specify your interpolation method and search criteria in order to provide faster and more accurate results (see *LIMIT*, below).

kDim is the interpolation criteria. If *kDim* = 2 or 0, two dimensional interpolation is applied (interpolate on a surface). If *kDim* = 3, three dimensional interpolation is applied (interpolate on a volume).

kOut specified how points outside of the domain are handled. If *kOut* = 0, use the value(s) of the nearest region point for points outside of the region. If *kOut* = 1, set results outside of the region to zero.

LIMIT specifies the number of nearby points considered for interpolation. The default is 20, and the minimum is 5. Lower values will reduce processing time; however, some distorted or irregular sets of points will require a higher *LIMIT* value to encounter three nodes for triangulation.

Output points are incorrect if they are not within the domain (area or volume) defined by the specified input points. Also, calculations for out-of-bound points require much more processing time than do points that are within bounds. Results mapping is available from the command line only.

INTP --

(***MOPER (p. 1147)**, *ParR*, *Par1*, INTP, *Par2*)

Finds the elements that contain each point in the array of *n* x 3 points in *Par1*. *Par2* will contain the set of element ID numbers and *ParR* will contain their *n* x 3 set of natural element coordinates (values between -1 and 1). *Par1* must be in global Cartesian coordinates.

SGET --

(***MOPER (p. 1147)**, *ParR*, *Par1*, SGET, *Par2*, *Label*, *Comp*)

Gets the nodal solution item corresponding to *Label* and *Comp* (see the **PLNSOL (p. 1425)** command) and interpolates it to the given element locations. *Par1* contains the *n* x 3 array of natural element coordinates (values between -1 and 1) of the *n* element ID numbers in *Par2*. *Par1* and *Par2* are usually the output of the ***MOPER (p. 1147)**,,,INTP operation. *ParR* contains the *n* interpolated results.

Val1*, *Val2*, . . . , *Val6

Additional input used in the operation. The meanings of *Val1* through *Val6* vary depending on the specified matrix operation. See the description of *Oper* for details.

Notes

Each starting array element number must be defined for each array parameter matrix if it does not start at the first location. For example, ***MOPER (p. 1147)**,A(2,3),B(1,4),MULT,C(1,5) multiplies submatrix B (starting at element (1,4)) by submatrix C (starting at element (1,5)) and puts the result in matrix A (starting at element (2,3)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The diagonal corner elements for each submatrix must be defined: the upper left corner by the array starting element (on this command), the lower right corner by the current values from the ***VCOL (p. 2077)** and ***VLEN (p. 2136)** commands. The default values are the (1,1) element and the last element in the matrix. No operations progress across matrix planes (in the 3rd dimension). Absolute values and scale factors may be applied to all parameters (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). Array elements should not be skipped with the ***VMASK (p. 2139)** and the *NINC* value of the ***VLEN (p. 2136)** specifications. See the ***VOPER (p. 2142)** command for details.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Matrix Operations

MOPT, *Lab*, *Value*

Specifies meshing options.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Meshing option to be specified (determines the meaning of *Value*):

AORDER

Mesh by ascending area size order. Set *Value* to ON to mesh smaller areas first. Using this results in finer meshes in critical areas for volume meshes; this can be used for cases where **SMRTSIZE (p. 1811)** does not mesh as needed. Default is OFF.

EXPND

Area mesh expansion (or contraction) option. (This option is the same as **SMRTSIZE (p. 1811)**,,,EXPND.) This option is used to size internal elements in an area based on the size of the elements on the area's boundaries.

Value is the expansion (or contraction) factor. For example, issuing **MOPT (p. 1151)**,EXPND,2 before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *Value* for this option should be greater than 0.5 but less than 4.

Value defaults to 1, which does not allow expansion or contraction of internal element sizes (except when using **AESIZE (p. 104)** sizing). If *Value* = 0, the default value of 1 will be used. The actual size of the internal elements will also depend on the TRANS option (or upon **AESIZE (p. 104)** or **ESIZE (p. 668)** sizing, if used).

TETEXPND

Tet-mesh expansion (or contraction) option. This option is used to size internal elements in a volume based on the size of the elements on the volume's boundaries.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value is the expansion (or contraction) factor. For example, issuing **MOPT (p. 1151)**,TET-EXPND,2 before meshing a volume will allow a mesh with elements that are approximately twice as large in the interior of the volume as they are on the boundary. If *Value* is less than 1, a mesh with smaller elements on the interior of the volume will be allowed. *Value* for this option should be greater than 0.1 but less than 3.

Value defaults to 1, which does not allow expansion or contraction of internal element sizes. If *Value* = 0, the default value of 1 will be used. If *Value* is greater than 2, mesher robustness may be affected.

The TETEXPND option is supported for both the **VMESH (p. 2140)** and **FVMESH (p. 784)** commands. Tet-mesh expansion is the only mesh control supported by **FVMESH (p. 784)**.

TRANS

Mesh-transition option. Controls how rapidly elements are permitted to change in size from the boundary to the interior of an area. (This option performs the same operation as **SMRTSIZE (p. 1811)**,,,,TRANS.)

Value is the transitioning factor. *Value* defaults to 2.0, which permits elements to approximately double in size as they approach the interior of the area. (If *Value* = 0, the default value of 2 will be used.) *Value* must be greater than 1 and, for best results, should be less than 4. The actual size of the internal elements will also depend on the EXPND option (or upon **AESIZE (p. 104)** or **ESIZE (p. 668)** sizing, if used).

For a quad mesh with any element size, this option has no effect, as the program strictly respects any face size to ensure the most uniform quad mesh possible. To obtain a graded mesh using this option, apply **LESIZE (p. 1041)** to the lines of the desired face.

AMESH

Triangle surface-meshing option. Valid inputs for *Value* are:

DEFAULT

Allows the program to choose which triangle mesher to use. In most cases, the program chooses the main triangle mesher, which is the Riemann space mesher. If the chosen mesher fails for any reason, the program invokes the alternate mesher and retries the meshing operation.

MAIN

The program uses the main triangle mesher (Riemann space mesher), and it does not invoke an alternate mesher if the main mesher fails. The Riemann space mesher is well suited for most surfaces.

ALTERNATE

The program uses the first alternate triangle mesher (3D tri-mesher), and it does not invoke another mesher if this mesher fails. This option is not recommended due to speed considerations. However, for surfaces with degeneracies in parametric space, this mesher often provides the best results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALT2

The program uses the second alternate triangle mesher (2D parametric space mesher), and it does not invoke another mesher if this mesher fails. This option is not recommended for use on surfaces with degeneracies (spheres, cones, etc.) or poorly parameterized surfaces because poor meshes may result.

QMESH

Quadrilateral surface meshing option. (Quadrilateral surface meshes will differ based on which triangle surface mesher is selected. This is true because all free quadrilateral meshing algorithms use a triangle mesh as a starting point.) Valid inputs for *Value* are:

DEFAULT

Let the program choose which quadrilateral mesher to use. In most cases, the program will choose the main quadrilateral mesher, which is the Q-Morph (quad-morphing) mesher. For very coarse meshes, the program may choose the alternate quadrilateral mesher instead. In most cases, the Q-Morph mesher results in higher quality elements. If either mesher fails for any reason, the program invokes the other mesher and retries the meshing operation. (Default.)

MAIN

The program uses the main quadrilateral mesher (Q-Morph mesher), and it does not invoke the alternate mesher if the main mesher fails.

ALTERNATE

The program uses the alternate quadrilateral mesher, and it does not invoke the Q-Morph mesher if the alternate mesher fails. To use the alternate quadrilateral mesher, you must also select **MOPT (p. 1151)**,AMESH,ALTERNATE or **MOPT (p. 1151)**,AMESH,ALT2.

VMESH

Tetrahedral element meshing option. Valid inputs for *Value* are:

DEFAULT

Let the program choose which tetrahedra mesher to use.

MAIN

Use the main tetrahedra mesher (Delaunay technique mesher). (GHS3D meshing technology by P. L. George, INRIA, France.) For most models, this mesher is significantly faster than the alternate mesher.

ALTERNATE

Use the alternate tetrahedra mesher (advancing front mesher). This mesher does not support the generation of a tetrahedral volume mesh from facets (**FVMESH (p. 784)**). If this mesher is selected and you issue the **FVMESH (p. 784)**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

command, the program uses the main tetrahedra mesher to create the mesh from facets and issues a warning message to notify you.

SPLIT

Quad splitting option for non-mapped meshing. If *Value* = 1, ON, or ERR, quadrilateral elements in violation of shape error limits are split into triangles (default). If *Value* = 2 or WARN, quadrilateral elements in violation of either shape error *or* warning limits are split into triangles. If *Value* = OFF, splitting does not occur, regardless of element quality.

LSMO

Line smoothing option. *Value* can be ON or OFF. If *Value* = ON, smoothing of nodes on area boundaries is performed during smoothing step of meshing. During smoothing, node locations are adjusted to achieve a better mesh. If *Value* = OFF (default), no smoothing takes place at area boundaries.

CLEAR

This option affects the element and node numbering after clearing a mesh. If *Value* = ON (default), the starting node and element numbers will be the lowest available number after the nodes and elements are cleared. If *Value* = OFF, the starting node and element numbers are not reset after the clear operation.

PYRA

Transitional pyramid elements option. *Value* can be ON or OFF. If *Value* = ON (default), the program automatically creates transitional pyramid elements, when possible. Pyramids may be created at the interface of tetrahedral and hexahedral elements, or directly from quadrilateral elements. For pyramids to be created, you must also issue the command **MSHAPE (p. 1188)**,1,3D (degenerate 3D elements). If *Value* = OFF, the program does not create transitional pyramid elements.

TIMP

Identifies the level of tetrahedra improvement to be performed when the next free volume meshing operation is initiated (**VMESH (p. 2140)**, **FVMESH (p. 784)**). (For levels 2-5, improvement occurs primarily through the use of face swapping and node smoothing techniques.) Valid inputs for *Value* are:

0

Turn off tetrahedra improvement. Although this value can lead to faster tetrahedral mesh creation, it is *not recommended* because it often leads to poorly shaped elements and mesh failures.

1

Do the minimal amount of tetrahedra improvement. (Default.) This option is supported by the main tetrahedra mesher only (**MOPT (p. 1151)**,VMESH,MAIN). If the alternate tetrahedra mesher (**MOPT (p. 1151)**,VMESH,ALTERNATE) is invoked with this setting, the program automatically performs tetrahedra improvement at level 3 instead (**MOPT (p. 1151)**,TIMP,3).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2

Perform the least amount of swapping/smoothing. No improvement occurs if all tetrahedral elements are within acceptable limits.

3

Perform an intermediate amount of swapping/smoothing. Some improvement is always done.

4

Perform the greatest amount of swapping/smoothing. Meshing takes longer with this level of improvement, but usually results in a better mesh.

5

Perform the greatest amount of swapping/smoothing, plus additional improvement techniques. This level of improvement usually produces results that are similar to those at level 4, except for very poor meshes.

6

For linear tetrahedral meshes, this value provides the same level of improvement as **MOPT (p. 1151)**,TIMP,5. For quadratic tetrahedral meshes, this value provides an additional pass of cleanup. This value is supported for both the main (**MOPT (p. 1151)**,VMESH,MAIN) and alternate (**MOPT (p. 1151)**,VMESH,ALTERNATE) tetrahedra meshers.

STAT

Display status of **MOPT (p. 1151)** settings. *Value* is ignored.

DEFA

Set all **MOPT (p. 1151)** options to default values. *Value* is ignored.

Value

Value, as described for each different *Lab* above.

Notes

See the *Modeling and Meshing Guide* for more information on the **MOPT (p. 1151)** command and its options.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesher Opts

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Area Cntrls

Main Menu>Preprocessor>Meshing>Size Cntrls>ManualSize>Global>Volu Cntrls

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options

Utility Menu>List>Status>Preprocessor>Solid Model

MORPH,*Option*,--,*Remeshopt*,*ElemSet*,*ARMAX*,*VOCH*,*ARCH*,*STEP*,*TIME*,*StrOpt*

Specifies morphing and remeshing controls.

PREP7 (p. 22): Morphing (p. 37)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

OFF

Turns off morphing for field elements (default).

ON

Turns on morphing for field elements.

--

Unused field

Remeshopt

OFF

Do not remesh (default).

ON

Remesh when element qualities fall below values specified by *ARMAX*, *VOCH*, or *ARCH* as explained below. Valid only when *Option* is ON.

ElemSet

ALL

Remesh all selected elements if the quality of the worst defined element falls below any quality requirement (default when *Remeshopt* = ON).

CompName

Specify a component name, up to 256 characters. All elements included in this component name are remeshed if the quality of the worst element falls below any quality requirement.

ARMAX

The maximum allowable element generalized aspect ratio. Defaults to 5.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VOCH

The maximum allowable change of element size (area or volume). Defaults to 3.

ARCH

The maximum allowable element aspect ratio change. Defaults to 3.

STEP

The frequency of element quality checking, based on time steps. A quality check takes place at the intervals defined by *STEP*. Defaults to 1 (quality check at every step).

TIME

A quality check takes place at the time point specified. Defaults to -1 (a quality check at every time point).

StrOpt

NO

There are no structural elements in the model (default).

YES

There are structural elements in the model and the morphing happens after the structural solution.

Notes

MORPH (p. 1156) is applicable to any non-structural field analysis. It activates displacement degrees of freedom for non-structural elements so that boundary conditions may be placed on the field mesh to constrain the movement of the non-structural mesh during morphing. It morphs the non-structural mesh using displacements transferred at the surface interface between the structural field and the non-structural field. The displacements of non-structural elements are mesh displacements to avoid mesh distortion, but have no physical meaning except at the interface. **MORPH (p. 1156)** does not support surface, link, or shell elements, or any element shape other than triangles, quads, tets, and bricks. Morphed fields must be in the global Cartesian system (**CSYS (p. 441)** = 0).

If *StrOpt* = YES, the following non-structural element types will be morphed:

- Acoustic: FLUID30, FLUID220, FLUID221, FLUID243, FLUID244,
- Electrostatic PLANE121, SOLID122, and SOLID123,
- Electric PLANE230, SOLID231, and SOLID232,
- Electromagnetic PLANE233, SOLID236, and SOLID237,
- Thermal PLANE35, PLANE55, PLANE77, PLANE292, PLANE293, SOLID70, SOLID87, SOLID90, SOLID278, SOLID279, and SOLID291,
- Diffusion PLANE238, SOLID239, and SOLID240,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Coupled-field PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227 with no structural degrees of freedom.

The following structural elements types are supported by *StrOpt* = YES:

- PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190,
- Coupled-field PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227 with structural degrees of freedom.

After each remesh, new databases and results files are written with the extensions `.rth0`*n* and `.db0`*n*, where *n* is the remesh file number (`FieldName.rth01`, `FieldName.rth02`, ... and `FieldName.db01`, `FieldName.db02`, etc.). The original database file is `FieldName.dbo`. The `FieldName.db01`, `FieldName.db02`, etc. files have elements that are detached from the solid model.

Remeshing has the following restrictions:

- Valid only for the electrostatic elements (PLANE121, SOLID122, and SOLID123)
- Limited to triangle (2D) and tetrahedral (3D) options of these elements
- No body loads allowed in the interior nodes of the remeshing domain
- Nodes on the boundary cannot be remeshed; remeshing will not work if morphing failed on the surface nodes
- Not suitable for extreme area or volume changes

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Element Morphing

Main Menu>Solution>Load Step Opts>Other>Element Morphing

MOVE, *NODE*, *KC1*, *X1*, *Y1*, *Z1*, *KC2*, *X2*, *Y2*, *Z2*

Calculates and moves a node to an intersection.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Move this node. If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

KC1

First coordinate system number. Defaults to 0 (global Cartesian).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X1, Y1, Z1

Input one or two values defining the location of the node in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R1, θ1, Z1 for cylindrical, or R1, θ1, Φ1 for spherical or toroidal.

KC2

Second coordinate system number.

X2, Y2, Z2

Input two or one value(s) defining the location of the node in this coordinate system. Input "U" for unknown value(s) to be calculated and input "E" to use an existing coordinate value. Fields are R2, θ2, Z2 for cylindrical, or R2, θ2, Φ2 for spherical or toroidal.

Notes

Calculates and moves a node to an intersection location. The node may have been previously defined (at an approximate location) or left undefined (in which case it is internally defined at the **SOURCE (p. 1824)** location). The actual location is calculated from the intersection of three surfaces (implied from three coordinate constants in two different coordinate systems). The three (of six) constants easiest to define should be used. The program will calculate the remaining three coordinate constants. All arguments, except *KC1*, must be input. Use the repeat command (***REPEAT (p. 1586)**) after the **MOVE (p. 1158)** command to define a line of intersection by repeating the move operation on all nodes of the line.

Surfaces of constant value are implied by some commands by specifying a single coordinate value. Implied surfaces are used with various commands (**MOVE (p. 1158)**, **KMOVE (p. 986)**, **NSEL (p. 1266)**, etc.). Three surfaces are available with each of the four coordinate system types. Values or X, Y, or Z may be constant for the Cartesian coordinate system; values of R, θ, or Z for the cylindrical system; and values of R, θ, Φ for the spherical and toroidal systems. For example, an X value of 3 represents the Y-Z plane (or surface) at X=3. In addition, the parameters for the cylindrical and spherical coordinate systems may be adjusted (**CS (p. 435)**, **LOCAL (p. 1059)**) to form elliptical surfaces. For surfaces in elliptical coordinate systems, a surface of "constant" radius is defined by the radius value at the X-axis. Surfaces of constant value may be located in local coordinate systems (**LOCAL (p. 1059)**, **CLOCAL (p. 352)**, **CS (p. 435)**, or **CSKP (p. 438)**) to allow for any orientation.

The intersection calculation is based on an iterative procedure (250 iterations maximum) and a tolerance of 1.0E-4. The approximate location of a node should be sufficient to determine a unique intersection if more than one intersection point is possible. Tangent "intersections" should be avoided. If an intersection is not found, the node is placed at the last iteration location.

This command is also valid in the **/MAP (p. 1107)** processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>To Intersect

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MP,*Lab*,*MAT*,*C0*,*C1*,*C2*,*C3*,*C4*

Defines a linear material property as a constant or a function of temperature.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1164)

Lab

Valid material property label. Applicable labels are listed under "Material Properties" in the input table for each element type in the *Element Reference*.

ALPD

Mass matrix multiplier for damping.

ALPX

Secant coefficients of thermal expansion (also ALPY, ALPZ).

BETD

Stiffness matrix multiplier for damping.

BETX

Coefficient of diffusion expansion (also BETY, BETZ)

BVIS

Bulk viscosity

C

Specific heat

CREF

Reference concentration (may not be temperature dependent)

CSAT

Saturated concentration

CTEX

Instantaneous coefficients of thermal expansion (also CTEY, CTEZ)

CVH

Heat coefficient at constant volume per unit of mass

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DENS

Mass density.

DMPR

Damping ratio.

DMPS

Constant structural damping coefficient.

DXX

Diffusivity coefficients (also DYY, DZZ)

EMIS

Emissivity. For default behavior, see Notes (p. 1164).

ENTH

Enthalpy. See Considerations for Enthalpy (p. 1164).

EX

Elastic moduli (also EY, EZ)

GXY

Shear moduli (also GYZ, GXZ)

HF

Convection or film coefficient

KXX

Thermal conductivities (also KYY, KZZ)

LSST

Electric loss tangent

LSSM

Magnetic loss tangent

MGXX

Magnetic coercive forces (also MGYY, MGZZ)

MURX

Magnetic relative permeabilities (also MURY, MURZ)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MU

Coefficient of friction

NUXY

Minor Poisson's ratios (also NUYZ, NUXZ) (NUXY = ν_{yx}, as described in Stress-Strain Relationships in the *Mechanical APDL Theory Reference*)

PERX

Electric relative permittivities (also PERY, PERZ)

Note:

If you enter permittivity values less than 1 for SOLID5, PLANE13, or SOLID98, the program interprets the values as absolute permittivity. Values input for PLANE222, PLANE223, SOLID225, SOLID226, or SOLID227 are always interpreted as relative permittivity.

PRXY

Major Poisson's ratios (also PRYZ, PRXZ) (PRXY = ν_{xy}, as described in Stress-Strain Relationships in the *Mechanical APDL Theory Reference*)

QRATE

Heat generation rate for thermal mass element MASS71. Fraction of plastic work converted to heat (Taylor-Quinney coefficient) or fraction of viscoelastic loss converted to heat for coupled-field elements PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227.

REFT

Reference temperature. Must be defined as a constant; C1 through C4 are ignored.

RH

Hall Coefficient.

RSVX

Electrical resistivities (also RSVY, RSVZ).

SBKX

Seebeck coefficients (also SBKY, SBKZ).

SONC

Sonic velocity.

THSX

Thermal strain (also THSY, THSZ).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VISC

Viscosity.

MAT

Material reference number to be associated with the elements (defaults to the current MAT setting (**MAT (p. 1111)**)).

C0

Material property value, or if a property-versus-temperature polynomial is being defined, the constant term in the polynomial. *C0* can also be a table name (%*tabname*%); if *C0* is a table name, *C1* through *C4* are ignored.

C1, C2, C3, C4

Coefficients of the linear, quadratic, cubic, and quartic terms, respectively, in the property-versus-temperature polynomial. Leave blank (or set to zero) for a constant material property.

Notes

MP (p. 1160) defines a linear material property as a constant or in terms of a fourth order polynomial as a function of temperature. (See the **TB (p. 1899)** command for nonlinear material property input.) Linear material properties typically require a single substep for solution, whereas nonlinear material properties require multiple substeps.

If the constants C1 - C4 are input, the polynomial

$$\text{Property} = C0 + C1(\text{T}) + C2(\text{T})^2 + C3(\text{T})^3 + C4(\text{T})^4$$

is evaluated at discrete temperature points with linear interpolation between points (that is, a piecewise linear representation) and a constant-valued extrapolation beyond the extreme points. First-order properties use two discrete points (±9999°). The **MPTEMP (p. 1178)** or **MPTGEN (p. 1179)** commands must be used for second and higher order properties to define appropriate temperature steps. To ensure that the number of temperatures defined via the **MPTEMP (p. 1178)** and **MPTGEN (p. 1179)** commands is minimally sufficient for a reasonable representation of the curve, Mechanical APDL generates an error message if the number is less than N, and a warning message if the number is less than 2N. The value N represents the highest coefficient used; for example, if C3 is nonzero and C4 is zero, a cubic curve is being used which is defined using 4 coefficients so that N = 4.

Some elements (for example, FLUID116) support tabular input for material properties. Use the ***DIM (p. 530)** command to create the table of property values as a function of the independent variables. Then input this table name (*C0* = %*tabname*%) when defining the property via the **MP (p. 1160)** command. Tabular material properties are calculated before the first iteration (that is, using initial values (**IC (p. 921)**)). For a list of elements that support tabular material properties and associated primary variables, see Defining Linear Material Properties Using Tabular Input in the *Material Reference*.

When defining a reference temperature (**MP (p. 1160)**,REFT), you can convert temperature-dependent secant coefficients of thermal expansion (SCTE) data from the definition temperature to the reference temperature. To do so, issue the **MPAMOD (p. 1166)** command.

This command is also valid in SOLUTION.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Considerations for Enthalpy (*Lab* = ENTH)

- To ensure correct results, you must define enthalpy over a large enough temperature range to span all computed temperatures during the solution. The **TB (p. 1899)** command does not extrapolate enthalpy values beyond the specified temp range like the **MP (p. 1160)** command does.
- If both the **TB (p. 1899)** and **MP (p. 1160)** commands are used to specify enthalpy values, enthalpy values defined via the **TB (p. 1899)** command are used and those defined via the **MP (p. 1160)** command are ignored.

Default behavior for Emissivity (*Lab* = EMIS)

There is no command default (p. 3) value for emissivity, and you must specify it by issuing **MP (p. 1160)**,EMIS. Otherwise, an error message appears. If you issue **MP (p. 1160)**,EMIS without specifying *C0*, *C1*, *C2*, *C3*, *C4* values, emissivity defaults to 0.

Product Restrictions

Command Option Lab	Available Products
ALPD	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ALPX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ALPY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ALPZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BETD	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BETX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BETY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BETZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BVIS	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CREF	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CSAT	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DXX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DYY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DZZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
C	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CTEX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CTEY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CTEZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CVH	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DENS	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DMPR	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DMPS	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
EMIS	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ENTH	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
EX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
EY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
EZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
GXY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
GXZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
GYZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
HF	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
KXX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
KYY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
KZZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
LSSM	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
LSST	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MGXX	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MGYY	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MGZZ	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MU	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MURX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MURY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MURZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
NUXY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
NUXZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
NUYZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PERX	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PERY	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PERZ	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PRXY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PRXZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PRYZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
QRAT	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
REFT	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
RH	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
RSVX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
RSVY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
RSVZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SBKX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SBKY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SBKZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SONC	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

THSX	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
THSY	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
THSZ	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
VISC	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPAMOD, *MAT*, *DEFTEMP*

Modifies temperature-dependent secant coefficients of thermal expansion.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Material number for which the secant coefficients of thermal expansion (SCTE's) are to be modified. Defaults to 1.

DEFTEMP

Definition temperature at which the existing SCTE-versus-temperature tables were defined. Defaults to zero.

Notes

This command converts temperature-dependent SCTE data (properties ALPX, ALPY, ALPZ) from the definition temperature (*DEFTEMP*) to the reference temperature defined by **MP (p. 1160)**,REFT or **TREF (p. 2013)**. If both the **MP (p. 1160)**,REFT and **TREF (p. 2013)** commands have been issued, the reference temperature defined by the **MP (p. 1160)**,REFT command will be used.

This command does not apply to the instantaneous coefficients of thermal expansion (properties CTEX, CTEY, CTEZ) or to the thermal strains (properties THSX, THSY, THSZ).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Convert ALPx

Main Menu>Preprocessor>Material Props>Convert ALPx

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Convert ALPx

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPCHG, *MAT*, *ELEM*

Changes the material number attribute of an element.

PREP7 (p. 22): Materials (p. 24)
SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Assign this material number to the element. Material numbers are defined with the material property commands (**MP (p. 1160)**).

ELEM

Element for material change. If ALL, change materials for all selected elements (**ESEL (p. 661)**).

Notes

Changes the material number of the specified element. Between load steps in SOLUTION, material properties cannot be changed from linear to nonlinear, nonlinear to linear, or from one nonlinear option to another.

If you change from one CHABOCHE model to another CHABOCHE model, the different models need to have the same number of data points.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Change Mat Num

Main Menu>Preprocessor>Material Props>Change Mat Num

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Change Mat Num

MPCOPY, - -, *MATF*, *MATT*

Copies linear material model data from one material reference number to another.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

- -

Unused field

MATF

Material reference number from where material property data will be copied.

MATT

Material reference number to where material property data will be copied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **MPCOPY (p. 1167)** command copies linear material properties only, which are all properties defined through the **MP (p. 1160)** command. If you copy a model that includes both linear and yield behavior constants (for example, a BKIN model), the **MPCOPY (p. 1167)** and **TBCOPY (p. 1959)**, ALL commands are used together to copy the entire model. All input data associated with the model is copied, that is, all data defined through the **MP (p. 1160)** and **TB (p. 1899)** commands.

Also, if you copy a material model using the Material Model Interface (**Edit> Copy**), both the commands **MPCOPY (p. 1167)** and **TBCOPY (p. 1959)**, ALL are issued, regardless of whether the model includes linear constants only, or if it includes a combination of linear and yield behavior constants.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPDATA, *Lab*, *MAT*, *SLOC*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*

Defines property data to be associated with the temperature table.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1171)

Lab

Valid property label. Applicable labels are listed under "Material Properties" in the input table for each element type in the *Element Reference*.

ALPD

Mass matrix multiplier for damping.

ALPX

Secant coefficients of thermal expansion (also ALPY, ALPZ). (See also **MPAMOD (p. 1166)** command for adjustment to reference temperature).

BETD

Stiffness matrix multiplier for damping.

BETX

Coefficient of diffusion expansion (also BETY, BETZ)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C

Specific heat.

CREF

Reference concentration (may not be temperature dependent)

CSAT

Saturated concentration

CTEX

Instantaneous coefficients of thermal expansion (also CTEY, CTEZ).

DENS

Mass density.

DMPS

Constant material damping coefficient.

DXX

Diffusivity coefficients (also DYY, DZZ)

EMIS

Emissivity.

ENTH

Enthalpy.

EX

Elastic moduli (also EY, EZ).

GXY

Shear moduli (also GYZ, GXZ).

HF

Convection or film coefficient.

KXX

Thermal conductivities (also KYY, KZZ).

LSST

Dielectric loss tangent.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MGXX

Magnetic coercive forces (also MGYY, MGZZ).

MU

Coefficient of friction.

MURX

Magnetic relative permeabilities (also MURY, MURZ).

NUXY

Minor Poisson's ratios (also NUYZ, NUXZ).

PERX

Electric relative permittivities (also PERY, PERZ).

PRXY

Major Poisson's ratios (also PRYZ, PRXZ).

QRATE

Heat generation rate.

REFT

Reference temperature (may not be temperature dependent).

RH

Hall Coefficient.

RSVX

Electrical resistivities (also RSVY, RSVZ).

SBKX

Seebeck coefficients (also SBKY, SBKZ).

SONC

Sonic velocity.

THSX

Thermal strain (also THSY, THSZ).

VISC

Viscosity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MAT

Material reference number to be associated with the elements (defaults to 1 if you specify zero or no material number).

SLOC

Starting location in table for generating data. For example, if *SLOC* = 1, data input in the *C1* field is the first constant in the table. If *SLOC* = 7, data input in the *C1* field is the seventh constant in the table, etc. Defaults to the last location filled + 1.

C1, C2, C3, . . . , C6

Property data values assigned to six locations starting with *SLOC*. If a value is already in this location, it is redefined. A blank (or zero) value for *C1* resets the previous value in *SLOC* to zero. A value of zero can only be assigned by *C1*. Blank (or zero) values for *C2* to *C6* leave the corresponding previous values unchanged.

Notes

Defines a table of property data to be associated with the temperature table. Repeat **MPDATA (p. 1168)** command for additional values (100 maximum). Temperatures must be defined first (**MPTEMP (p. 1178)**). Also stores assembled property function table (temperature and data) in virtual space.

This command is also valid in SOLUTION.

Product Restrictions

Ansys Mechanical Enterprise

The command **MPDATA (p. 1168)**,LSST is only available to the Ansys Mechanical Enterprise product family (Ansys Mechanical Enterprise, Ansys Mechanical Enterprise PrepPost, and Ansys Mechanical Enterprise Solver).

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPDELE, *Lab*, *MAT1*, *MAT2*, *INC*, *LCHK*

Deletes linear material properties.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Material property label (see **MP (p. 1160)** command for valid labels). If ALL, delete properties for all applicable labels.

MAT1*, *MAT2*, *INC

Delete materials from *MAT1* to *MAT2* (defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1* = ALL, *MAT2* and *INC* are ignored and the properties for all materials are deleted.

LCHK

Specifies the level of element-associativity checking:

NOCHECK

No element-associativity check occurs. This option is the default.

WARN

When a section, material, or real constant is associated with an element, Mechanical APDL issues a message warning that the necessary entity has been deleted.

CHECK

The command terminates, and no section, material, or real constant is deleted if it is associated with an element.

Notes

This command is also valid in SOLUTION.

The *LCHK* argument is valid only when *Lab* = ALL.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPDRES, *LabF*, *MATF*, *LabT*, *MATT*

Reassembles existing material data with the temperature table.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabF

Material property label associated with *MATF*.

MATF

Material reference number of property to restore from virtual space.

LabT

Material property label associated with *MATT* (defaults to label associated with *MATF*).

MATT

Material reference number assigned to generated property (defaults to *MATF*).

Notes

Restores into the database (from virtual space) a data table previously defined (**MP (p. 1160)**) for a particular property, assembles data with *current* database temperature table, and stores back in virtual space as a new property.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Other>Change Mat Props>Modify Temps

Main Menu>Preprocessor>Material Props>Modify Temps

Main Menu>Solution>Other>Change Mat Props>Modify Temps

/MPLIB, *R-W_opt*, *PATH*

Sets the default material library read and write paths.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

R-W_opt

Determines what path is being set. Possible values are:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

READ

Set the read path.

WRITE

Set the write path.

STAT

Report what read and write paths are currently in use.

PATH

The directory path to be used for material library files.

Notes

The **/MPLIB (p. 1173)** command sets two path strings used in conjunction with the material library feature and the **MPREAD (p. 1176)** and **MPWRITE (p. 1181)** commands.

For **MPREAD (p. 1176)**, when you use the *LIB* option and no directory path is given in the file name, the command searches for the file in these locations: the current working directory, the user's home directory, the user-specified material library directory (as defined by the **/MPLIB (p. 1173)**,READ,*PATH* command), and */ansys_dir*/matlib.

For **MPWRITE (p. 1181)**, when you use the *LIB* option and the directory portion of the specification for the material library file is blank, the command writes the material library file to the directory specified by the **/MPLIB (p. 1173)**,WRITE,*PATH* command (if that path has been set). If the path has not been set, the default is to write the file to the current working directory.

The Material Library files supplied with the distribution media are meant for demonstration purposes only. These files are not intended for use in customer applications.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Lib Path Status

Main Menu>Preprocessor>Material Props>Material Library>Lib Path Status

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Lib Path Status

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPLIST, *MAT1*, *MAT2*, *INC*, *Lab*, *TEVL*

Lists linear material properties.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT1*, *MAT2*, *INC

List materials from *MAT1* to *MAT2* (defaults to *MAT1*) in steps of *INC* (defaults to 1). If *MAT1*= ALL (default), *MAT2* and *INC* are ignored and properties for all material numbers are listed.

Lab

Material property label (see the **MP (p. 1160)** command for labels). If ALL (or blank), list properties for all labels. If EVLT, list properties for all labels evaluated at TEVL.

TEVL

Evaluation temperature for *Lab* = EVLT listing (defaults to **BFUNIF (p. 259)**).

Notes

For *Lab* = EVLT, when the property is from tables, the **MPPLOT (p. 1175)** command will not be valid because the property could be a function of more than temperature.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Properties>All Materials

Utility Menu>List>Properties>All Matls, All Temps

Utility Menu>List>Properties>All Matls, Specified Temp

Utility Menu>List>Properties>Specified Matl, All Temps

MPPLOT, *Lab*, *MAT*, *TMIN*, *TMAX*, *PMIN*, *PMAX*

Plots linear material properties as a function of temperature.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Linear material property label (EX, EY, etc.) (**MP (p. 1160)**).

MAT

Material reference number. Defaults to 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TMIN

Minimum abscissa value to be displayed.

TMAX

Maximum abscissa value.

PMIN

Minimum property (ordinate) value to be displayed.

PMAX

Maximum property value.

Notes

When the property is from tables, the **MPPLOT (p. 1175)** command will not be valid because the property could be a function of more than temperature.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Materials

MPREAD,*Fname*,*Ext*,--,LIB

Reads a file containing material properties.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify the LIB option, the default directory is the current working directory. If you specify the LIB option, the default is the following search path: the current working directory, the user's home directory, MPLIB_DIR (as specified by the **/MPLIB (p. 1173)**,READ,PATH command) and */ansys_dir/*matlib (as defined by installation). If you use the default for your directory, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

If you omit the default extension is MP. extension is units_MPL, where units is the system of units currently in use. (See the description of the **/UNITS (p. 2054)** command.) For example, if **/UNITS (p. 2054)** is set to SI, the extension defaults to SI_MPL.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

--

Unused field.

LIB

Reads material library files previously written with the **MPWRITE (p. 1181)** command. (See the description of the LIB option for the **MPWRITE (p. 1181)** command.) The only allowed value for LIB is LIB.

The LIB field indicates that the specified file was written by **MPWRITE (p. 1181)** using the LIB option, and that the file is consistent with the material library file format. When the **MPREAD (p. 1176)** command executes, Mechanical APDL reads material properties defined in the specified file into the current Mechanical APDL working database. The currently selected material, as defined by the **MAT (p. 1111)** command (**MAT (p. 1111)**,MAT), determines the material number used when reading the material properties. The LIB option for **MPREAD (p. 1176)** and **MPWRITE (p. 1181)** supports storing and retrieving both linear and nonlinear properties.

Notes

Material properties written to a file without the LIB option do not support nonlinear properties. Also, properties written to a file without the LIB option are restored in the same material number as originally defined. To avoid errors, use **MPREAD (p. 1176)** with the LIB option only when reading files written using **MPWRITE (p. 1181)** with the LIB option.

If you omit the LIB option for **MPREAD (p. 1176)**, this command supports only linear properties.

Material numbers are hardcoded. If you write a material file without specifying the LIB option, then read that file in via the **MPREAD (p. 1176)** command with the LIB option, Mechanical APDL does not write the file to a new material number; instead, it writes the file to the old material number (the number specified on the **MPWRITE (p. 1181)** command that created the file.)

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Export Library

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Import Library

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Select Units

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Read from File

Main Menu>Preprocessor>Material Props>Material Library>Export Library

Main Menu>Preprocessor>Material Props>Material Library>Import Library

Main Menu>Preprocessor>Material Props>Material Library>Select Units

Main Menu>Preprocessor>Material Props>Read from File

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Export Library

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Import Library

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Select Units

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Read from File

---

MPRINT, *KEY*

Specifies that radiation matrices are to be printed.

AUX12 (p. 62): Radiation Matrix Method (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Print key:

0

Do not print matrices.

1

Print matrices.

Command Default

Matrices are not printed.

Notes

Specifies that the element and node radiation matrices are to be printed when the **WRITE (p. 2185)** command is issued. If *KEY* = 1, form factor information for each element will also be printed.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Write Matrix

MPTEMP, *SLOC*, *T1*, *T2*, *T3*, *T4*, *T5*, *T6*

Defines a temperature table for material properties.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SLOC

Starting location in table for entering temperatures. For example, if *SLOC* = 1, data input in the *T1* field applies to the first constant in the table. If *SLOC* = 7, data input in the *T1* field applies to the seventh constant in the table, etc. Defaults to the last location filled + 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T1, T2, T3, . . . , T6

Temperatures assigned to six locations starting with *SLOC*. If a value is already in this location, it will be redefined. A blank (or zero) value for *T1* resets the previous value in *SLOC* to zero. A value of zero can only be assigned by *T1*. Blank (or zero) values for *T2* to *T6* leave the corresponding previous values unchanged.

Command Default

No temperature table defined (that is, properties must be defined as a constant or linear function of temperature via the **MP (p. 1160)** command).

Notes

Defines a temperature table to be associated with the property data table (**MPDATA (p. 1168)**). These temperatures are also used for polynomial property evaluation, if defined (**MP (p. 1160)**). Temperatures must be defined in non-descending order. Issue **MATER (p. 1112)** $ **STAT (p. 1861)** to list the current temperature table. Repeat **MPTEMP (p. 1178)** command for additional temperatures (100 maximum). If all arguments are blank, the temperature table is erased.

For clear definition, the temperature range you define with the **MPTEMP (p. 1178)** command should include the entire range you'll use in subsequently defined materials. To assist the user in this, the first (and only the first) excursion out of the temperature range defined by the **MPTEMP (p. 1178)** commands is flagged with a warning message. Similarly, the reference temperature (**TREF (p. 2013)** or **MP (p. 1160)**,REFT commands) should also fall in this same temperature range. If not and **MP (p. 1160)**,ALPX was used, a note will be output. If not, and **MP (p. 1160)**,CTEX or **MP (p. 1160)**,THSX was used, an error message will be output.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

MPTGEN,*STLOC*,*NUM*,*TSTRT*,*TINC*

Adds temperatures to the temperature table by generation.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

STLOC

Starting location in table for generating temperatures. Defaults to last location filled + 1.

NUM

Number of temperatures to be generated (1-100).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TSTRT

Temperature assigned to *STLOC* location.

TINC

Increment previous temperature by *TINC* and assign to next location until all *NUM* locations are filled.

Notes

Adds temperatures to the temperature table by generation. May be used in combination (or in place of) the **MPTEMP (p. 1178)** command.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Other>Change Mat Props>Generate Temp

Main Menu>Preprocessor>Material Props>Generate Temp

Main Menu>Solution>Other>Change Mat Props>Generate Temp

MPTRES, *Lab*, *MAT*

Restores a temperature table previously defined.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Material property label (**MP (p. 1160)**).

MAT

Material reference number.

Notes

Restores into the database (from virtual space) a temperature table previously defined (**MP (p. 1160)**) for a particular property. The existing temperature table in the database is erased before this operation.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Other>Change Mat Props>Restore Temps

Main Menu>Preprocessor>Material Props>Restore Temps

Main Menu>Solution>Other>Change Mat Props>Restore Temps

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MPWRITE, *Fname*, *Ext*, --, LIB, *MAT*

Writes linear material properties in the database to a file (if the LIB option is not specified) or writes both linear and nonlinear material properties (if LIB is specified) from the database to a file.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including directory). If you do not specify the LIB option, the default directory is the current working directory. If you specify LIB and you have specified a material library directory (via the **/MPLIB (p. 1173)** command), that directory is the default. Otherwise, the default is the current working directory. If you use the default for your directory, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

If you omit the LIB option, the default extension is MP. If you specify the LIB option, the default extension is units_MPL, where units is the system of units currently in use. (See the description of the **/UNITS (p. 2054)** command.) For example, if **/UNITS (p. 2054)** is set to BIN, the extension defaults to BIN_MPL.

--

Unused field.

LIB

The only value allowed for this field is the string "LIB."

The LIB option indicates that you wish to have properties associated with the material (MAT) written to the specified material library file using the material library file format. The material library file format is ASCII-text-based Mechanical APDL command input. Certain commands associated with this format have been modified to interpret the string "_MATL" to mean the currently selected material. This feature makes the material library file independent of the material number in effect when the file was written; this enables you to restore the properties into the Mechanical APDL database using the material number of your choice. The LIB option also enables you to save both linear and nonlinear properties. If you omit the LIB option, you can save linear properties only.

MAT

Specifies the material to be written to the named material library file. *There is no default; you must either specify a material or omit the MAT argument.* Even if you specify a *MAT* value, Mechanical APDL ignores it if the LIB argument is not specified.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Writes linear material properties currently in the database to a file. The file is rewound before and after writing.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Export Library

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Import Library

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Library>Select Units

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Write to File

Main Menu>Preprocessor>Material Props>Material Library>Export Library

Main Menu>Preprocessor>Material Props>Material Library>Import Library

Main Menu>Preprocessor>Material Props>Material Library>Select Units

Main Menu>Preprocessor>Material Props>Write to File

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Export Library

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Import Library

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Library>Select Units

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Write to File

---

/MREP, *NAME*, *ARG1*, *ARG2*, *ARG3*, . . . , *ARG4*, *ARG5*, *ARG6*, *ARG7*, *ARG8*, *ARG9*, *ARG10*, *ARG11*, *ARG12*, *ARG13*, *ARG14*, *ARG15*, *ARG16*, *ARG17*, *ARG18*

Enables you to reissue the graphics command macro "name" during a replot or zoom operation.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NAME

The name identifying the macro file or macro block on a macro library file. The name can contain up to eight characters maximum and must begin with a letter.

ARG1, ARG2, ARG3, . . . , ARG18

Values to be passed into the file or block.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command reissues the graphics command macro "name" during a replot operation (**/REPLOT (p. 1587)**) or a zoom (**/ZOOM (p. 2203)**) operation. The program passes the command macro arguments to the replot and zoom feature for use by the graphics macro. You should place the **s-MREP (p. 1182)** command at the end of the graphics command macro, following the last graphics command within the macro, to enable the replot or zoom feature.

Menu Paths

This command cannot be accessed from a menu.

MRPM, *VAL1*

Defines the revolutions per minute (RPM) for a machine rotation.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1

The RPM value (no default).

Notes

A different RPM value can be defined at each load step. The RPM value is used to postprocess the equivalent radiated power from the structural surface (the **PRAS (p. 1464)** and **PLAS (p. 1387)** commands) or the radiated sound power level (the **PRFAR (p. 1490)** and **PLFAR (p. 1412)** commands).

Menu Paths

This command cannot be accessed from a menu.

MSAVE, *Key*

Sets the solver memory saving option. This option only applies to the PCG solver (including PCG Lanczos).

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Activation key:

0 or OFF

Use global assembly for the stiffness matrix (and mass matrix, when using PCG Lanczos) of the entire model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 or ON

Use an element-by-element approach when possible to save memory during the solution. In this case, the global stiffness (and mass) matrix is not assembled; element stiffness (and mass) is regenerated during PCG or PCG Lanczos iterations.

Notes

MSAVE (p. 1183),ON only applies to and is the default for parts of the model using the following element types with linear material properties that meet the conditions listed below.

- SOLID186 (Structural Solid only)
- SOLID187

The following conditions must also be true:

- The PCG solver has been specified.
- Small strains are assumed (**NLGEOM (p. 1231)**,OFF).
- No prestress effects (**PSTRES (p. 1551)**) are included.
- All nodes on the supported element types must be defined (that is, the midside nodes cannot be removed using the **EMID (p. 624)** command).
- For elements with thermally dependent material properties, **MSAVE (p. 1183)**,ON applies only to elements with uniform temperatures prescribed.
- The default element coordinate system must be used.

If you manually force **MSAVE (p. 1183)**,ON by including it in the input file, the model can include the following additional conditions:

- The analysis can be a modal analysis using the PCG Lanczos method (**MODOPT (p. 1140)**,LANPCG).
- Large deflection effects (**NLGEOM (p. 1231)**,ON) can be included for SOLID186 and/or SOLID187 elements.
- SOLID185 (brick shapes and KEYOPT(2) = 3 only) elements can be included for small strains (**NLGEOM (p. 1231)**,OFF).

All other element types or other parts of the model that don't meet the above criteria will be solved using global assembly (**MSAVE (p. 1183)**,OFF). This command can result in memory savings of up to 70 percent over the global assembly approach for the part of the model that meets the criteria. Depending on the hardware (for example, processor speed, memory bandwidth, etc.), the solution time may increase or decrease when this feature is used.

This memory-saving feature runs in parallel when multiple processors are used with the **/CONFIG (p. 410)** command or in a distributed-memory parallel (DMP) solution. The gain in performance with using multiple processors with this feature turned on should be similar to the default case when this feature is turned off. Performance also improves when using the uniform reduced integration option for SOLID186 elements.

This command does not support the *layered* option of the SOLID185 and SOLID186 elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **MSAVE (p. 1183)** command is not valid in these circumstances:

- When the PCG solver is used in harmonic analyses.
- When the matrix is unsymmetric.

When using **MSAVE (p. 1183)**,ON with the **PCGOPT (p. 1365)** command, note the following restrictions:

- For static and modal analyses, **MSAVE (p. 1183)**,ON is not valid when using a *Lev_Diff* value of 5 on the **PCGOPT (p. 1365)** command; *Lev_Diff* is automatically reset to 2.
- For modal analyses, **MSAVE (p. 1183)**,ON is not valid with the *StrmCk* option of the **PCGOPT (p. 1365)** command; *Strmck* is set to OFF.
- For all analysis types, **MSAVE (p. 1183)**,ON is not valid when the Lagrange multiplier option (*LM_Key*) of the **PCGOPT (p. 1365)** command is set to ON; the **MSAVE (p. 1183)** activation key is set to OFF.
- For linear perturbation static and modal analyses, **MSAVE (p. 1183)**,ON is not valid; the **MSAVE (p. 1183)** activation key is set to OFF.
- For static analyses, **MSAVE (p. 1183)**,ON is not valid when the *Fallback* option of the **PCGOPT (p. 1365)** command is enabled; *Fallback* is automatically reset to OFF.

When using **MSAVE (p. 1183)**,ON for modal analyses, no `.full` file will be created. The `.full` file may be necessary for subsequent analyses (for example, harmonic, transient mode-superposition, or spectrum analyses). To generate the `.FULL` file, rerun the modal analysis using the **WRFULL (p. 2184)** command.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

***MSG**, *Lab*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*

Writes an output message via the Mechanical APDL message subroutine.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Lab

Label for output and termination control:

INFO --

Writes the message with no heading (default).

NOTE --

Writes the message with a "NOTE" heading.

WARN --

Writes the message with a "WARNING" heading. Also writes the message to the errors file, `Jobname.err`.

ERROR --

Writes the message with a "ERROR" heading and causes run termination (if batch) at earliest "clean exit" point. Also writes the message to the errors file, `Jobname.err`.

FATAL --

Writes the message with a "FATAL ERROR" heading and causes run termination immediately. Also writes the message to the errors file, `Jobname.err`.

UI --

Writes the message with a "NOTE" heading and displays it in the message dialog box. This option is most useful in GUI mode.

VAL1*, *VAL2*, *VAL3*, . . . , *VAL8

Numeric or alphanumeric character values to be included in message. Values may be the results of parameter evaluations. All numeric values are assumed to be double precision. The FORTRAN nearest integer (NINT) function is used to form integers for the %I specifier.

Notes

Allows writing an output message via the Mechanical APDL message subroutine. Also allows run termination control. This command is used only when contained in a prepared file read into the Mechanical APDL program (that is, ***USE (p. 2059)**,**/INPUT (p. 948)**, etc.). A message format must immediately follow the ***MSG (p. 1185)** command (on a separate line, without parentheses, as described below).

The message format may be up to 80 characters long, consisting of text strings and predefined "data descriptors" between the strings where numeric or alphanumeric character data are to be inserted. The normal descriptors are %I for integer data, %G for double precision data, %C for alphanumeric character data, and %/ for a line break. The corresponding FORTRAN data descriptors are I9, 1PG16.9 and A8, respectively. Each descriptor must be preceded by a blank. There must be one data descriptor for each specified value (8 maximum) in the order of the specified values.

Enhanced descriptions may also be used:

%w.pE	w is field width
%w.pG	p is precision
%w.pF	
%%	a single percent sign
%wC; %wS	character string

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

%-wC; %-wS	left justify string
%wX	w blank characters
%wI	integer format
%0wI	pad integer with leading zeros rather than blanks
%0w.pI	w is field width; p is number of characters filled

Do not begin ***MSG (p. 1185)** format lines with ***IF (p. 927)**, ***ELSE (p. 620)**, ***ELSEIF (p. 620)**, or ***ENDIF (p. 636)**. If the last nonblank character of the message format is an ampersand (&), a second line will also be read as a continuation of the format. Up to nine continuations (ten total lines) may be read. If normal descriptions are used, then consecutive blanks are condensed into one blank upon output, and a period is appended. Up to ten lines of output of 72 characters each may be produced (using the %/ descriptor). Two examples follow.

Here is an example of the ***MSG (p. 1185)** command and a format to print a message with two integer values and one real value:

```
*MSG, INFO, 'Inner',25,1.2,148
Radius ( %C) = %I, Thick = %G, Length = %I
```

The output line is:

```
Radius (Inner) = 25, Thick = 1.2, Length = 148.
```

Here is an example illustrating multiline displays in GUI message windows:

```
*MSG,UI,Vcoilrms,THTAv,Icoilrms,THTAi,Papprnt,Pelec,PF,indctnc
Coil RMS voltage, RMS current, apparent pwr, actual pwr, pwr factor: %/&
Vcoil = %G V (electrical angle = %G DEG) %/&
Icoil = %G A (electrical angle = %G DEG) %/&
APPARENT POWER = %G W %/&
ACTUAL POWER = %G W %/&
Power factor: %G %/&
Inductance = %G %/&
VALUES ARE FOR ENTIRE COIL (NOT JUST THE MODELED SECTOR)
```

Note:

The **/UIS (p. 2045)**,MSGPOP command controls which messages are displayed in the message dialog box when the GUI is active. All messages produced by the ***MSG (p. 1185)** command are subject to the **/UIS (p. 2045)** specification, with one exception, If *Lab* = UI, the message will be displayed in the dialog box regardless of the **/UIS (p. 2045)** specification.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MSHAPE, *KEY*, *Dimension*

For elements that support multiple shapes, specifies the element shape to be used for meshing.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Key indicating the element shape to be used:

0

Mesh with quadrilateral-shaped elements when *Dimension* = 2D mesh with hexahedral-shaped elements when *Dimension* = 3D.

1

Mesh with triangle-shaped elements when *Dimension* = 2D mesh with tetrahedral-shaped elements when *Dimension* = 3D.

Dimension

Specifies the dimension of the model to be meshed:

2D

2D model (area mesh).

3D

3D model (volume mesh).

Command Default

Since specification of element shape (**MSHAPE (p. 1188)**) and meshing type (**MSHKEY (p. 1190)**) are so closely related, the element shape that Mechanical APDL meshes with depends on the combination of the values that are set for the two commands. The table below explains what happens when you fail to specify values for these settings.

Your action...	How it affects the mesh...
You issue the **MSHAPE (p. 1188)** command with no arguments.	The program uses quadrilateral-shaped or hexahedral-shaped elements to mesh the model, depending on whether you are meshing an area or a volume.
You do not specify an element shape, but you do specify the type of meshing to be used (**MSHKEY (p. 1190)**).	The program uses the default shape of the element to mesh the model. It uses the type of meshing that you specified.
You specify neither an element shape nor the type of meshing to be used.	The program uses the default shape of the element to mesh the model. It uses

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Your action...	How it affects the mesh...
	whichever type of meshing is the default for that shape.

Notes

If no value is specified for *Dimension* the value of *KEY* determines the element shape that will be used for both 2D and 3D meshing. In other words, if you specify **MSHAPE (p. 1188)**,0, quadrilateral-shaped and hexahedral-shaped elements will be used. If you specify **MSHAPE (p. 1188)**,1, triangle-shaped and tetrahedral-shaped elements will be used.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided

Main Menu>Preprocessor>Meshing>Mesher Opts

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options

---

MSHCOPY, *KEYLA*, *LAPTRN*, *LACOPY*, *KCN*, *DX*, *DY*, *DZ*, *TOL*, *LOW*, *HIGH*

Simplifies the generation of meshes that have matching node element patterns on two different line groups (in 2D) or area groups (3D).

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEYLA

Copy line mesh (default) if LINE, 0 or 1. Copy area mesh if AREA, or 2.

LAPTRN

Meshed line/area to be copied, or a component name containing a list. If *LAPTRN* = P, graphical picking is enabled (valid only in the GUI).

LACOPY

Unmeshed line/area to get copied mesh, or a component name containing a list. If *LACOPY* = P, graphical picking is enabled (valid only in the GUI).

KCN

In coordinate system *KCN*, *LAPTRN* + *DX DY DZ* = *LACOPY*.

DX, DY, DZ

Node location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, Dθ, DΦ for spherical or toroidal).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TOL

Tolerance. Defaults to 1.e--4.

LOW

Name of low node component to be defined (optional).

HIGH

Name of high node component to be defined (optional).

Notes

Matching meshes are used for rotational (cyclic) symmetry, or for contact analysis using coupling or node-to-node gap elements. See Using CPCYC and MSHCOPY Commands in the *Modeling and Meshing Guide* for more information.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Area Mesh

Main Menu>Preprocessor>Modeling>Copy>Line Mesh

MSHKEY, *KEY*

Specifies whether free meshing or mapped meshing should be used to mesh a model.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Key indicating the type of meshing to be used:

0

Use free meshing (the default).

1

Use mapped meshing.

2

Use mapped meshing if possible; otherwise, use free meshing. If you specify **MSHKEY (p. 1190)**,2, SmartSizing will be inactive even while free meshing non-map-meshable areas.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

As stated above, free meshing is the default. However, since the **MSHKEY (p. 1190)** and **MSHAPE (p. 1188)** settings are closely related, you should refer to the table in the **MSHAPE (p. 1188)** command description for more information about defaults.

Notes

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Areas>Mapped>3 or 4 sided

Main Menu>Preprocessor>Meshing>Mesh>Areas>Target Surf

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided

Main Menu>Preprocessor>Meshing>Mesher Opts

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options

---

MSHMID, *KEY*

Specifies placement of midside nodes.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Key indicating placement of midside nodes:

0

Midside nodes (if any) of elements on a region boundary follow the curvature of the boundary line or area (the default).

1

Place midside nodes of all elements so that element edges are straight. Allows coarse mesh along curves.

2

Do not create midside nodes (elements will have removed midside nodes).

Notes

This command is also valid for rezoning.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesher Opts

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options

---

MSHPATTERN, *KEY*

Specifies pattern to be used for mapped triangle meshing.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Key indicating triangle pattern to be used (the figures below illustrate the pattern that will be used for each value of *KEY*):

0

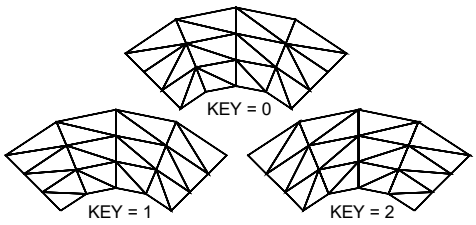
Allow Mechanical APDL choose the pattern (default). The program maximizes the minimum angle of the triangular-shaped elements that are created.

1

Unidirectional split at node I.

2

Unidirectional split at node J.



Notes

"Mapped triangle meshing" refers to the Mechanical APDL program's ability to take a map-meshable area and mesh it with triangular elements, based on the value of **MSHPATTERN (p. 1192)**,*KEY*. This type of meshing is particularly useful for analyses that involve the meshing of rigid contact elements.

This command is valid only when you have specified that Mechanical APDL use triangle-shaped elements (**MSHAPE (p. 1188)**,1,2D)--or you are meshing with an element that supports only triangles--*and* you have specified mapped meshing (**MSHKEY (p. 1190)**,1) to mesh an area.

For details about mapped meshing with triangles, see the *Modeling and Meshing Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesher Opts

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Global Meshing Options

---

MSOLVE, *NUMSLV*, *VAL1*, *VAL2*, *Lab*, *ANGFIX*

Starts multiple solutions for an acoustic analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUMSLV

Number of multiple solutions (load steps). This number corresponds to the number of random samplings for the diffuse sound field in a random acoustic analysis, or the incident angles of the plane wave when the Floquet periodic boundary condition is present. Default = 1.

VAL1

The meaning of *VAL1* depends on the *Lab* value.

For *Lab* = DSF, *VAL1* is the norm convergence tolerance defined by comparing the averaged radiated diffuse sound power of two multiple sampling sets over the frequency range for the diffuse sound field (default = 0.05).

For *Lab* = APHI or ATHETA, *VAL1* is the beginning angle for the incident angle sweep of the plane wave (default = 0).

VAL2

The meaning of *VAL2* depends on the *Lab* value.

For *Lab* = DSF, *VAL2* is the interval of the norm convergence check for the diffuse sound field (default = 5).

For *Lab* = APHI or ATHETA, *VAL2* is the ending angle for the incident angle sweep of the plane wave (default = 0).

Lab

Label indicating the type of acoustic analysis:

DSF

Diffuse sound field with multiple solutions (default).

APHI

Plane wave angle sweep with fixed ϕ angle and varied θ angle (see the **APORT (p. 170)** command).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ATHETA

Plane wave angle sweep with fixed θ angle and varied ϕ angle (see the **APORT (p. 170)** command).

ANGFIX

The value of the fixed incident angle for the plane wave angle sweep (used only when *Lab* = APHI or ATHETA).

Notes

The **MSOLVE (p. 1193)** command starts multiple solutions (load steps) for a random acoustic analysis with multiple samplings or for the angle sweep of the incident plane wave with the Floquet periodic boundary condition, as described below.

Random Acoustic Analysis (*Lab* = DSF)

Use *Lab* = DSF for a random acoustic analysis. The process is controlled by the norm convergence tolerance (*VAL1*) or the number of multiple solutions (*NUMSLV*) if the solution steps reach the defined number.

The program checks the norm convergence by comparing two averaged sets of radiated sound powers with the interval *VAL2* over the frequency range. For example, if *VAL2* = 5, the averaged values from 5 solutions are compared with the averaged values from 10 solutions, then the averaged values from 10 solutions are compared with the averaged values from 15 solutions, and so on.

The incident diffuse sound field is defined via the **DFSWAVE (p. 529)** command.

The average result of multiple solutions with different samplings is calculated via the **PRAS (p. 1464)** or **PLAS (p. 1387)** command.

Plane Wave Incident Angle Sweep (*Lab* = APHI or ATHETA)

Use *Lab* = APHI or ATHETA to perform an angle sweep for the incident plane wave defined by the **APORT (p. 170)** command. The process is controlled by the number of multiple solutions (*NUMSLV*). The plane wave port must be defined with the default values of incident angles prior to the **MSOLVE (p. 1193)** command.

The sound power parameters are calculated over the sweeping angles during postprocessing by the **PRAS (p. 1464)** or **PLAS (p. 1387)** command.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MSOPT, *Option*, *Sname*, *Value1*, *Value2*, *Value3*, *Value4*, *Value5*, *Value6*, *Value7*

Specifies solution options for a multistage cyclic symmetry analysis.

PREP7 (p. 22): Special Purpose (p. 37)
SOLUTION (p. 38): Dynamic Options (p. 40)
POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Multistage cyclic symmetry analysis option. There is no default. You must choose one of the following options:

CSYS

Activates a previously defined cyclic coordinate system by the reference number specified in *Value1*. *Sname* is ignored. This option is only valid in the /PREP7 processor.

Value1

Cylindrical coordinate system reference number. You must have already created the coordinate system by issuing prior commands like **CS (p. 435)** or **LOCAL (p. 1059)** to define it. Defaults to 1 where the global Cartesian Z axis is the cyclic symmetry axis.

Note:

You must define the coordinate system before defining the stages.

NEW

Creates a new stage with the name entered in *Sname* and the *Value1* - *Value6* specifications listed in the table below. This option is only valid in the /PREP7 processor.

Sname

An alphanumeric name used to identify the stage. *Sname* may be up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. Names beginning with an underscore (for example, _LOOP) are reserved for use by Mechanical APDL and should be avoided. The component name ALL is not permitted.

Specifications for a new stage created via MSOPT (p. 1195) (*Option* = NEW):	
Argument	**Description**
Value1	The number of sectors.
Value2	The harmonic index.
Value3	The name of a component[a] containing all base sector elements (defaults to '_*Sname*_BASE_ELM').

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value4	The name of a component[a] containing all base sector nodes (defaults to '*_Sname*_BASE_NOD').
Value5	The name of a component[a] containing all duplicate sector elements (defaults to '*_Sname*_DUPL_ELM)'. The duplicate sector[b] is not included in the analysis if this argument is blank and no '*_Sname*_DUPL_ELM' component exists.
Value6	The name of a component[a] containing all duplicate sector nodes (defaults to '*_Sname*_DUPL_NOD'). The duplicate sector[b] is not included in the analysis if this argument is blank and no '*_Sname*_DUPL_NOD' component exists.
Value7	The name of the lead harmonic stage if the stage being defined is a member of a multiharmonic group and is not the lead harmonic. **Note:** A multiharmonic group is composed of a single lead harmonic stage and one or more secondary harmonic stages formed by stage cloning. To properly set up the multiharmonic group, you must specify the name of the lead harmonic stage using *Value7* for all secondary harmonic stages or stage clones. Leave *Value7* blank if you are issuing **MSOPT (p. 1195)**,NEW to define: • the lead harmonic stage in a multiharmonic group • a stage that is not part of a multiharmonic group.

[a] Up to 256 characters (see **CM (p. 356)** command).

[b] A duplicate sector can always be present in the model even if it is not required in the analysis. For a more detailed discussion on duplicate sectors in multistage analyses, see Requirements for Stage Meshes in the *Multistage Cyclic Symmetry Analysis Guide*.

Important:

When creating a new stage, the base and duplicate sector meshes should be coincident. The offset between the number of base and duplicate coincident nodes is constant throughout the whole model. The same is true for the element numbers.

DELETE

Deletes the stage identified by *Sname*.

Sname

The name of the stage to be deleted. Entities such as nodes and elements contained in the stage are unaffected. Only the grouping and the related constraint equations are concerned.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EXPAND

Specifies stages (identified by *Sname*) and sectors (sector number specified in *Value1*) for subsequent expansion. This option is only valid in the /POST1 processor.

Sname

The name of the stage to be expanded. A value of 0 resets all expansion settings. A value of ALL means all existing stages will be expanded (default).

Value1

The sector number. A value of 0 resets all sector settings. A value of ALL means all sectors will be expanded (default).

LIST

Lists the stage identified by *Sname* with the level of detail specified in *Value1*.

Sname

The name of the stage to be listed. If blank, list all stages (default).

Value1

Key for specifying the level of detail.

0 (*or* OFF)

Basic listing (default).

1 (*or* ON)

Detailed listing, including constraint equations information. Note that the interstage constraint equations number information is only listed for the stage with the smallest number of sectors.

MODIFY

Sets the harmonic index of a stage identified by *Sname* to the integer specified in *Value1*.

Sname

The name of the stage to be modified.

Value1

The new harmonic index. Existing cyclic and multistage interface constraint equations will be deleted.

RESET

Deletes all stages and resets all multistage analysis settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **MSOPT (p. 1195)** command is used to specify solution options for a multistage cyclic symmetry analysis. It is not part of the **CYCLIC (p. 462)** procedure for a cyclic symmetry analysis.

When you issue **MSOPT (p. 1195)**,EXPAND, subsequent **SET (p. 1724)** commands read the data set from the specified `.rst` file and expand the nodes and elements to the stages and sectors specified via **MSOPT (p. 1195)**,EXPAND.

Example Usage

Example: Static Analysis of a Compressor Model with 4 Axial Stages Without a Duplicate Sector

Example: Linear Perturbation Modal Analysis of a Simplified Model with 2 Axial Stages and a Non-planar Interstage Boundary

Example: Modal Analysis of Turbomachinery Stage Modeled as 2 Radial Stages with Offset Cyclic Edge Starting Points

Example: Mutistage Multiharmonic Modal Analysis of a Hollow Cylinder Modeled Using 2 Stages

Example: Multiharmonic Linear Perturbation Modal Analysis of a Simplified Model with 3 Axial Stages

Menu Paths

This command cannot be accessed from a menu.

/MSTART, *Label*, *KEY*

Controls the initial GUI components.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Label identifying the GUI component:

ZOOM

Pan, Zoom, Rotate dialog box, off by default.

WORK

Offset Working Plane dialog box, off by default.

WPSET

Working Plane Settings dialog box, off by default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ABBR

Edit Toolbar/Abbreviations dialog box, off by default.

PARM

Scalar Parameters dialog box, off by default.

SELE

Select Entities dialog box, off by default.

ANNO

Annotation dialog box, off by default.

HARD

Hard Copy dialog box, off by default.

UTIL

Activates the pre-6.1 (UIDL) GUI (off by default).

KEY

Switch value:

OFF or 0

Component does not appear when GUI is initialized.

ON or 1

Component appears when GUI is initialized.

Command Default

Same as *Label* defaults.

Notes

Controls which components appear when the Graphical User Interface (GUI) is initially brought up. This command is valid only *before* the GUI is brought up (**/MENU (p. 1121)**,ON) and is intended to be used in the `start.ans` file. It only affects how the GUI is *initialized*; you can always bring up or close any component once you are in the GUI.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MSTOLE,*METHOD*,*Namesurf*,*Namefluid*

Adds two extra nodes from FLUID116 elements to SURF151 or SURF152 elements for convection analyses.

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

METHOD

Mapping method:

0

Hybrid method (default).

1

Projection method.

2

Minimum centroid distance method.

Namesurf

Component name for a group of SURF151 or SURF152 elements. The component name must be 32 characters or less, and it must be enclosed in single quotes (for example, 'COM152') when the **MSTOLE (p. 1200)** command is manually typed in.

Namefluid

Component name for a group of FLUID116 elements. The component name must be 32 characters or less, and it must be enclosed in single quotes (for example, 'COM116') when the **MSTOLE (p. 1200)** command is manually typed in.

Notes

For convection analyses, the **MSTOLE (p. 1200)** command adds two extra nodes from FLUID116 elements to SURF151 or SURF152 elements by employing the specified mapping method. In the hybrid method, the projection method is tried first and if it fails the centroid distance method is used. The SURF151 or SURF152 elements and the FLUID116 elements must be grouped into components and named using the **CM (p. 356)** command.

The SURF151 or SURF152 extra node option must be set for two extra nodes (KEYOPT(5) = 2).

For more information, see Using the Surface Effect Elements in the *Thermal Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*MULT, *M1*, *T1*, *M2*, *T2*, *M3*

Performs the matrix multiplication M3 = $M1^{(T1)}$*$M2^{(T2)}$.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

M1

Name of matrix *M1*. Must have been previously specified by a ***DMAT (p. 551)** or ***SMAT (p. 1801)** command.

T1

Transpose key. Set *T1* = TRANS to use the non-conjugate transpose of *M1*. Set *T1* = CTRANS to use the conjugate transpose of *M1*. CTRANS is only applicable when the *M1* matrix is complex. If blank, transpose will not be used.

M2

Name of matrix *M2*. Must have been previously specified by a ***DMAT (p. 551)** command.

T2

Transpose key. Set *T2* = TRANS to use the non-conjugate transpose of *M2*. Set *T2* = CTRANS to use the conjugate transpose of *M2*. CTRANS is only applicable when the *M2* matrix is complex. If blank, transpose will not be used.

M3

Name of resulting matrix, *M3*. Must be specified.

Notes

The matrices must be dimensionally consistent such that the number of columns of *M1* (or the transposed matrix, if requested) is equal to the number of rows of *M2* (or the transposed matrix, if requested).

You cannot multiply two sparse matrices with this command (that is, *M1* and *M2* cannot both be sparse). The resulting matrix, *M3*, will always be a dense matrix, no matter what combination of input matrices is used (dense*sparse, sparse*dense, or dense*dense).

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*MWRITE,*ParR*,*Fname*,*Ext*,--,*Label*,*n1*,*n2*,*n3*

Writes a matrix to a file in a formatted sequence.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the array parameter. See ***SET (p. 1720)** for name restrictions.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

If the file name fields are left blank, the default file is the current output file.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Label

Can use a value of IJK, IKJ, JIK, JKI, KIJ, KJI, or blank (JIK).

n1, n2, n3

Write as (((*ParR*(i,j,k), k = 1,n1), i = 1, n2), j = 1, n3) for *Label* = KIJ. *n1*, *n2*, and *n3* default to the corresponding dimensions of the array parameter ParR.

Notes

Writes a matrix or vector to a specified file in a formatted sequence. You can also use the ***VWRITE (p. 2174)** command to write data to a specified file. Both commands contain format descriptors on the line immediately following the command. The format descriptors can be in either FORTRAN or C format.

FORTRAN format descriptors are enclosed in parentheses. They must immediately follow the ***MWRITE (p. 1202)** command on a separate line of the same input file. The word FORMAT should not be included. The format must specify the number of fields to be written per line, the field width, the placement of the decimal point, etc. There should be one field descriptor for each data item written. The write operation uses the available system FORTRAN FORMAT conventions (see your system FORTRAN manual). Any standard FORTRAN *real* format (such as (4F6.0), (E10.3,2X,D8.2), etc.) and character format (A) may be used. Integer (I) and list-directed (*) descriptors may *not* be used. Text may be included in

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the format as a quoted string. *The FORTRAN descriptor must be enclosed in parentheses* and the format must not exceed 80 characters (including parentheses).

The "C" format descriptors are used if the first character of the format descriptor line is not a left parenthesis. "C" format descriptors may be up to 80 characters long, consisting of text strings and predefined "data descriptors" between the strings where numeric or alphanumeric character data are to be inserted. The normal descriptors are %I for integer data, %G for double precision data, %C for alphanumeric character data, and %/ for a line break. There must be one data descriptor for each specified value in the order of the specified values. The enhanced formats described in ***MSG (p. 1185)** can also be used.

The starting array element number must be defined. Looping continues in the directions indicated by the Label argument. The number of loops and loop skipping may also be controlled via ***VLEN (p. 2136)** and ***VMASK (p. 2139)**, which work in the *n2* direction (by row on the output file), and by ***VCOL (p. 2077)** , which works in the *n1* direction (by column in the output file). The vector specifications ***VABS (p. 2072)** and ***VFACT (p. 2092)** apply to this command, while ***VCUM (p. 2080)** does not apply to this command. See ***VOPER (p. 2142)** for details. If you are in the GUI, the ***MWRITE (p. 1202)** command must be contained in an externally prepared file and read into Mechanical APDL (that is, ***USE (p. 2059)**, **/INPUT (p. 948)**, etc.).

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Parameters>Write to File

MXPAND,*NMODE*,*FREQB*,*FREQE*,*Elcalc*,*SIGNIF*,*MSUPkey*,*ModeSelMethod*,*EngCalc*

Specifies modal or buckling analysis expansion options.

SOLUTION (p. 38): Dynamic Options (p. 40)
SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NMODE

Number of modes or array name (enclosed in percent signs) to expand and write. If blank or ALL, expand and write all modes within the frequency range specified. If -1, do not expand and do not write modes to the results file during the analysis. If an array name is input, the array must contain 1 for the expanded modes and zero otherwise, where the array index corresponds to the mode number. To specify an array containing the individual modes to expand, enclose the array name in percent (%) signs (for example, **MXPAND (p. 1203)**,%arrname%). Use the ***DIM (p. 530)** command to define the array.

FREQB

Beginning, or lower end, of frequency range of interest. If *FREQB* and *FREQE* are both blank, expand and write the number of modes specified without regard to the frequency range. Defaults to the entire range.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FREQE

Ending, or upper end, of frequency range of interest.

Elcalc

Element calculation key:

NO

Do not calculate element results, reaction forces, and energies (default).

YES

Calculate element results, reaction forces, energies, and the nodal degree of freedom solution.

SIGNIF

Expand only those modes whose significance level exceeds the *SIGNIF* threshold (only applicable when *ModeSelMethod* is defined).

If *ModeSelMethod* = MODC, the significance level of a mode is defined as the mode coefficient divided by the maximum mode coefficient of all modes.

If *ModeSelMethod* = EFFM, the significance level of a mode is defined as the modal effective mass, divided by the total mass.

If *ModeSelMethod* = DDAM, the significance level of a mode is defined as the modal effective weight, divided by the total weight.

Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not expanded. The higher the *SIGNIF* threshold, the fewer the number of modes expanded. *SIGNIF* defaults to 0.001, except for the case of DDAM mode selection method where it defaults to 0.01. If *SIGNIF* is specified as 0.0, it is taken as 0.0.

MSUPkey

Element result superposition key:

NO

Do not write element results to the mode file `Jobname.mode`.

YES

Write element result to the mode file for use in the expansion pass of a subsequent mode-superposition PSD, spectrum, transient, or harmonic analysis. This value is the default if all of the following conditions exist:

- *Elcalc* = YES.
- The mode shapes are normalized to the mass matrix.
- *FREQB* and *FREQE* are blank or 0.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- No superelement is defined.

ModeSelMethod

Methods for mode selection (not supported for complex eigensolvers):

blank

No mode selection is performed (default).

EFFM

The mode selection is based on the modal effective masses.

MODC

The mode selection is based on the mode coefficients.

DDAM

The mode selection is based on DDAM procedure (see Mode Selection Based on the DDAM Method in the *Structural Analysis Guide* for more information).

EngCalc

Additional element energies calculation key:

NO

Do not calculate additional energies (default).

YES

Calculate average, amplitude, and peak values for the following: stiffness and kinetic energies, and damping energy.

Command Default

Mode shapes (displacements) are expanded and output to the results file (so that if you need to inspect the mode shapes, it is not necessary to run the analysis again). Element stresses are not expanded.

Notes

Specifies the number of modes to expand and write over a frequency range for a modal (**ANTYPE (p. 162)**,MODAL) or buckling (**ANTYPE (p. 162)**,BUCKLE) analysis. If used in SOLUTION, this command is valid only within the first load step.

There is no limit on the number of expanded modes (*NMODE*). However, there *is* a limit on the maximum number of modes used via the ***GET (p. 797)**,,MODE (p. 831) command, mode combinations, and the **MDAMP (p. 1117)** command.

With *MSUPkey* = YES, the computed element results (*Elcalc* = YES) are written to `Jobname.mode` for use in subsequent downstream mode-superposition analyses, including harmonic, transient, and all spectrum analyses. This significantly reduces computation time for the combination or expansion passes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For limitations and available elemental results, see Option: Number of Modes to Expand (**MXPAND**) in the *Structural Analysis Guide*.

The calculation of additional energies (*EngCalc* = YES) is valid only for *Method* = DAMP on the **MODOPT (p. 1140)** command and *Method* = QRDAMP with *Cpxmod* = CPLX on the **MODOPT (p. 1140)** command. The average, amplitude, and peak potential energies for complex results (see Potential Energy (or Stiffness Energy) for Complex Results) calculated for a linear perturbation full harmonic analysis with a nonlinear base analysis are approximate solutions and may not be accurate.

If a mode selection method (*ModeSelMethod*) is defined, only the selected modes will be expanded. See Using Mode Selection in the *Structural Analysis Guide* for more details about the procedure.

For array input (*NMODE*), the array must be dimensioned to be the size of the number of modes extracted (*NMODE* on the **MODOPT (p. 1140)** command). A value of 1 in the array indicates the mode is to be expanded, and a value of 0 indicates not to expand the mode. For the DAMP modal solution, the modes are in pairs, so be sure to verify that both modes of a pair have the same value. (For example, if modes #3 and #4 are a pair, indices 3 and 4 in the array should have the same value, 0 or 1.)

For linear perturbation modal analyses, you must set both *Elcalc* and *MSUPkey* to YES so that the downstream stress expansion pass can produce a solution consistent with the linear or nonlinear base (static or full transient) analysis. The prestressed nonlinear element history (saved variables) is accessible only in the first and second phases of the linear perturbation. The downstream mode superposition analysis (such as MSUP or PSD) can only reuse the nonlinear information contained in the `Jobname.mode` file that is generated in the linear perturbation.

In a distributed-memory parallel (DMP) analysis, you must issue **MXPAND (p. 1203)** to specify the number of modes to expand when computing the modes and mode shapes. In a DMP run, **MXPAND (p. 1203)** cannot be issued in an expansion pass (**EXPASS (p. 714)**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Expand Modes

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Expand Modes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N Commands

N, *NODE*, *X*, *Y*, *Z*, *THXY*, *THYZ*, *THZX*

Defines a node.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node number to be assigned. A previously defined node of the same number will be redefined. Defaults to the maximum node number used +1.

X, Y, Z

Node location in the active coordinate system (R, θ, Z for cylindrical, R, θ, Φ for spherical or toroidal). If *X* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

THXY

First rotation about nodal Z (positive X toward Y).

THYZ

Second rotation about nodal X (positive Y toward Z).

THZX

Third rotation about nodal Y (positive Z toward X).

Notes

Defines a node in the active coordinate system (**CSYS (p. 441)**). The nodal coordinate system is parallel to the global Cartesian system unless rotated. Rotation angles are in degrees and redefine any previous rotation angles. See the **NMODIF (p. 1247)**, **NANG (p. 1208)**, **NROTAT (p. 1264)**, and **NORA (p. 1252)** commands for other rotation options.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>In Active CS

Main Menu>Preprocessor>Modeling>Create>Nodes>On Working Plane

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NANG, *NODE, X1, X2, X3, Y1, Y2, Y3, Z1, Z2, Z3*

Rotates a nodal coordinate system by direction cosines.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Rotate coordinate system of this node.

X1, X2, X3

Global X, Y, Z components of a unit vector in new nodal X direction.

Y1, Y2, Y3

Global X, Y, Z components of a unit vector in new nodal Y direction.

Z1, Z2, Z3

Global X, Y, Z components of a unit vector in new nodal Z direction.

Notes

Rotates a nodal coordinate system to the orientation specified by the X, Y and Z direction cosines. Existing rotation specifications on the node are redefined. If only two of the three unit vectors are specified, the third is defined according to the right hand rule. It is the responsibility of the user to ensure that input direction cosines are orthogonal in a right-handed system.

See the **NMODIF (p. 1247)**, **NROTAT (p. 1264)**, and **NORA (p. 1252)** commands for other rotation options.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>By Vectors

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>By Vectors

NAXIS, *Action, Val*

Generates nodes for general axisymmetric element sections.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Specifies one of the following command behaviors:

GEN

Generates nodes around the axis of an axisymmetric section (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CLEAR

Clears all nodes around the axis of an axisymmetric section.

EFACET

Specifies the number of facets per edge between nodal planes and integration planes in the circumferential direction to display using PowerGraphics. This option is only valid with **/ESHAPE (p. 665)**,1 and **RSYS (p. 1639)**,SOLU commands.

Val

Tolerance value *or* number of facets per edge:

TOLER

When *Action* = GEN, the tolerance to use for merging the generated nodes around the axis.

NUM

When *Action* = EFACET, the number of facets per element edge for element plots:

AUTO

Use program-chosen facets per edge (default).

1

Use 1 facet per edge (default for elements with 9, 10, 11, or 12 nodal planes). Shows nodal and integration planes only.

2

Use 2 facets per edge (default for elements with 5, 6, 7, or 8 nodal planes, and maximum for elements with 9, 10, 11, or 12 nodal planes).

3

Use 3 facets per edge (default for elements with 3 or 4 nodal planes, and maximum for elements with 6, 7, or 8 nodal planes).

4

Use 4 facets per edge (maximum for elements with 5 nodal planes).

5

Use 5 facets per edge (maximum for elements with 4 nodal planes).

6

Use 6 facets per edge (maximum for elements with 3 nodal planes).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **NAXIS (p. 1208)** command generates or clears the nodes for general axisymmetric element sections. The command applies to elements SURF159, SOLID272, and SOLID273.

The generate option (*Action* = GEN) operates automatically on any current-technology axisymmetric element. Any nodes within the tolerance value (*TOLER*) of the axis are merged into a single node. The default tolerance is 1.0e-4.

If you want to change the number of nodes, use the clear option (*Action* = CLEAR) before regenerating the nodes.

To cause the 3D element plot to appear more like the actual 3D model, use **NAXIS (p. 1208)**,EFACET,*NUM*, where *NUM* > 1. In this case, the coordinate system specified for displaying element and nodal results (RSYS) must be solution (**RSYS (p. 1639)**,SOLU); otherwise, Ansys resets *NUM* to 1.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>About AXIS Section

---

NCNV, *KSTOP*, *DLIM*, *ITLIM*, *ETLIM*, *CPLIM*

Sets the key to terminate an analysis.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KSTOP

Program behavior upon nonconvergence:

0

Do not terminate the analysis if the solution fails to converge.

1

Terminate the analysis and the program execution if the solution fails to converge (default).

2

Terminate the analysis, but not the program execution, if the solution fails to converge.

DLIM

Terminates program execution if the largest nodal DOF solution value (displacement, temperature, etc.) exceeds this limit. Defaults to 1.0E6 for all DOF except MAG and A. Defaults to 1.0E10 for MAG and A.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ITLIM

Terminates program execution if the cumulative iteration number exceeds this limit (defaults to infinity).

ETLIM

Terminates program execution if the elapsed time (seconds) exceeds this limit (defaults to infinity).

CPLIM

Terminates program execution if the CPU time (seconds) exceeds this limit (defaults to infinity).

Command Default

As described for each argument above.

Notes

Sets the key to terminate an analysis if not converged, or if any of the following limits are exceeded for nonlinear and full transient analyses: DOF (displacement), cumulative iteration, elapsed time, or CPU time limit. Applies only to static and transient analyses (**ANTYPE (p. 162)**,STATIC and **ANTYPE (p. 162)**,TRANS). Time limit checks are made at the end of each equilibrium iteration.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Criteria to Stop

Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL

Main Menu>Solution>Load Step Opts>Nonlinear>Criteria to Stop

NDELE,*NODE1*,*NODE2*,*NINC*

Deletes nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2*, *NINC

Delete nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and all selected nodes (**NSEL (p. 1266)**) are deleted. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Deletes selected nodes that are not connected to elements. Nodes may also be redefined instead of deleted, if desired. Boundary conditions (displacements, forces, etc.) as well as any coupling or constraint equations containing the deleted nodes are also deleted.

This command is also valid in the **/MAP (p. 1107)** processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

Main Menu>Preprocessor>Modeling>Delete>Nodes

NDIST, *ND1*, *ND2*

Calculates and lists the distance between two nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ND1

First node in distance calculation. If *ND1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

ND2

Second node in distance calculation.

Notes

NDIST (p. 1212) lists the distance between nodes *ND1* and *ND2*, as well as the current coordinate system offsets from *ND1* to *ND2*, where the X, Y, and Z locations of *ND1* are subtracted from the X, Y, and Z locations of *ND2* (respectively) to determine the offsets. **NDIST (p. 1212)** is valid in any coordinate system except toroidal (**CSYS (p. 441)**,3).

NDIST (p. 1212) returns a variable, called "*_RETURN*," which contains the distance value. You can use this value for various purposes, such as the calculation of distributed loads. In interactive mode, you can access this command by using the Model Query Picker (**Utility Menu> List> Picked Entities**), where you can also access automatic annotation functions and display the value on your model.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>ND distances

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NDSURF, *Snode*, *Telem*, *DIMN*

Generates surface elements overlaid on the edge of existing elements and assigns the extra node as the closest fluid element node.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Snode

Component name for the surface nodes of the solid elements. The component name must be 32 characters or less.

Telem

Component name for the target fluid elements. The component name must be 32 characters or less.

DIMN

Model dimensionality:

2

2D model.

3

3D model.

Notes

This command macro is used to generate surface effect elements (SURF151 or SURF152) overlaid on the surface of existing plane or solid elements and, based on proximity, to determine and assign the extra node (FLUID116) for each surface element. The active element type must be SURF151 or SURF152 with appropriate settings for KEYOPT(4), KEYOPT(5), KEYOPT(6), and KEYOPT(8).

The surface nodes of the plane or solid elements must be grouped into a node component and the fluid elements must be grouped into an element component and named using the **CM (p. 356)** command. The names must be enclosed in single quotes (for example, 'NOD') when the **NDSURF (p. 1213)** command is manually typed in.

When using the GUI method, node and element components are created through the picking dialog boxes associated with this command.

The macro is applicable for the SURF151, SURF152, and FLUID116 element types.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Area to Fluid

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Line to Fluid

Main Menu>Preprocessor>Modeling>Create>Elements>Surf / Contact>Surf Effect>Attch to Fluid>Node to Fluid

---

NEQIT,*NEQIT*,*FORCEkey*

Specifies the maximum number of equilibrium iterations for nonlinear analyses.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NEQIT

Maximum number of equilibrium iterations allowed each substep.

FORCEkey

One iteration forcing key:

FORCE

Forces one iteration per substep. Leave this field blank otherwise.
Using one iteration per substep may result in unconverged solutions for nonlinear analysis, and the program may not indicate divergence in this case. This option is intended primarily for use by the Ansys Workbench interface. Keep in mind that forcing one iteration per substep is only recommended under very specific conditions; for example, nonlinearity in bonded penalty type contact. Under these conditions the solution typically converges in one iteration.

Command Default

The maximum number of equilibrium iterations per substep is set between 15 and 26, depending on the physics of the problem.

Notes

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Equilibrium Iter

Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Solution>Load Step Opts>Nonlinear>Equilibrium Iter

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/NERR, *NMERR*, *NMABT*, - -, *IFKEY*, *NUM*

Limits the number of warning and error messages displayed.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NMERR

Maximum number of warning and error messages displayed per command. Defaults to 5 for interactive runs with the GUI turned on, 20 for interactive runs with the GUI turned off, 200 for batch runs. If *NMERR* is negative, the absolute value of *NMERR* is used as the maximum number of warning and error messages written to the error file (`Jobname.err`) per command, as well as the maximum number of messages displayed per command.

NMABT

Maximum number of warning and error messages allowed per command before run aborts (must be greater than zero). Maximum value is 99,999,999. Defaults to 10,000.

- -

Unused field.

IFKEY

Specifies whether or not to abort if an error occurs during a **/INPUT (p. 948)** operation:

0 *or* OFF

Do not abort. This option is the default.

1 *or* ON

Abort.

NUM

The number of invalid command warnings before a stop warning will be issued:

0

Disables the stop warning/error function.

n

An integer value representing the number of warnings that will be encountered before prompting the user to stop (default = 5). The first error encountered will ALWAYS result in a prompt.

Note:

Invalid command warnings and error tracking are mutually exclusive.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

As described above.

Notes

Limits the number of warning and error messages displayed for any one command in an interactive run.

Warning and error messages continue to be written to `Jobname.err` regardless of these limits, unless *NMERR* is negative. There is no way to totally suppress writing of warning and error messages to the error file.

Issue this command with *NUM* = *n* to specify the number of "invalid command" warnings to be encountered before the user is prompted to stop. You can then continue or abort the run. If you choose to abort the run, the log file can be saved so that any of the processing up to that point can be appended to an input that rectifies the condition. A batch run always aborts on the first error. Issue **/NERR (p. 1215)**,STAT to list current settings.

Issue **/NERR (p. 1215)**,DEFA to reset values to initial defaults.

An *IFKEY* value of 1 or ON causes Mechanical APDL to terminate immediately upon encountering an error during a file **/INPUT (p. 948)** operation. However, use of this option may cause the following conditions to occur:

- The **/INPUT (p. 948)** command may abort if issued for a log file (*jobname*`.log`).
- Some macros may terminate.
- A CAD connection may fail after reading only a small portion of a CAD model.

The command is valid in any processor.

Menu Paths

Utility Menu>MenuCtrls>Message Controls

NFORCE, *ITEM*

Sums the nodal forces and moments of elements attached to nodes.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITEM

Specifies the selected set of nodes for summing forces and moments for contact elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(blank)

Sums the nodal forces of elements for all selected nodes and excludes contact elements (elements 169-177).

CONT

Sums the nodal forces of elements for contact nodes only.

BOTH

Sums the nodal forces of elements for all selected nodes, including contact elements.

Notes

Sums and prints, in each component direction for each selected node, the nodal force and moment contributions of the selected elements attached to the node. If all elements are selected, the sums are usually zero except where constraints or loads are applied. The nodal forces and moments may be displayed (**/PBC (p. 1356)**,FORC and **/PBC (p. 1356)**,MOME). Use **PRESOL (p. 1481)** to print nodal forces and moments on an element-by-element basis. You can use the **FORCE (p. 772)** command to specify which component (static, damping, inertia, or total) of the nodal load is to be used. Nodal forces associated with surface loads are not included.

This vector sum is printed in the global Cartesian system. Moment summations are about the global origin unless another point is specified with the **SPOINT (p. 1836)** command. The summations for each node are printed in the global Cartesian system unless transformed (**RSYS (p. 1639)**). This command is generally not applicable to axisymmetric models because moment information from the NFORCE command is not correct for axisymmetric elements.

Selecting a subset of elements (**ESEL (p. 661)**) and then issuing this command will give the forces and moments required to maintain equilibrium of that set of elements. The effects of nodal coupling and constraint equations are ignored. The option *ITEM* = CONT provides the forces and moments for the contact elements (CONTA172, CONTA174, CONTA175, and CONTA177). Setting *ITEM* = BOTH provides the forces and moments for all selected nodes, including contact elements.

This command also includes the **FSUM (p. 780)** command function which vectorially sums and prints, in each component direction for the total selected node set, the nodal force and moment contributions of the selected elements attached to the selected node set.

Using NFORCE (p. 1216) in a Spectrum or PSD Analysis (ANTYPE (p. 162), SPECTR)

When using **NFORCE (p. 1216)** in a spectrum analysis (after the combination file has been input through **/INPUT (p. 948)**,,MCOM and when **SPOPT (p. 1837)** has not been issued with *Elcalc* = YES during the spectrum analysis), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS (p. 1639)** does not apply. The forces are in the nodal coordinate systems, and the vector sum is always printed in the global coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The spectrum analysis summation is available when the element results are written to the mode file, Jobname.mode (*MSUPkey* = Yes on the **MXPAND (p. 1203)** command).

Because modal displacements cannot be used to calculate contact element nodal forces, *ITEM* does not apply to spectrum and PSD analyses.

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Sum @ Each Node

NGEN, *ITIME*, *INC*, *NODE1*, *NODE2*, *NINC*, *DX*, *DY*, *DZ*, *SPACE*

Generates additional nodes from a pattern of nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME*, *INC

Do this generation operation a total of *ITIME* times, incrementing all nodes in the given pattern by *INC* each time after the first. *ITIME* must be > 1 for generation to occur.

NODE1*, *NODE2*, *NINC

Generate nodes from the pattern of nodes beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and the pattern is all selected nodes (**NSEL (p. 1266)**). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

DX*, *DY*, *DZ

Node location increments in the active coordinate system (DR, Dθ, DZ for cylindrical, DR, Dθ, DΦ for spherical or toroidal).

SPACE

Spacing ratio. Ratio of last division size to first division size. If > 1.0, divisions increase. If < 1.0, divisions decrease. Ratio defaults to 1.0 (uniform spacing).

Note:

The *average* spacing ratio remains 1.0, such that the location of the last generated set will be the same regardless of *SPACE*. *SPACE* only serves to skew the position of the nodes between the pattern set and the last set.

Notes

Generates additional nodes from a given node pattern. Generation is done in the active coordinate system. Nodes in the pattern may have been generated in any coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid in the **/MAP (p. 1107)** processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Nodes>Copy

NKPT, *NODE*, *NPT*

Defines a node at an existing keypoint location.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Arbitrary reference number for node. If zero or blank, defaults to the highest node number +1 (**NUMSTR (p. 1301)**).

NPT

Keypoint number defining global X, Y, Z location. If *NPT* = All, then a node will be placed at each selected keypoint. If *NPT* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NPT*.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>On Keypoint

NLADAPTIVE, *Component*, *Action*, *Criterion*, *Option*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*

Defines the criteria under which the mesh is refined or modified during a nonlinear solution.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Component

Specifies the element component upon which this command should act:

ALL

All selected components, or all selected elements if no component is selected (default).

Name

Component name.

Action

Action to perform on the selected component(s):

ADD

Add a criterion to the database.

LIST

List the criteria defined for the specified component(s).

DELETE

Delete the criteria defined for the specified component(s).

ON

Enable the defined criteria for the specified component(s) and specify how frequently and when to check them (via ON,,,*VAL1*,*VAL2*,*VAL3*):

VAL1 -- Checking frequency. If > 0, check criteria at every *VAL1* substeps. If < 0, check criteria at each of the *VAL1* points (approximately equally spaced) between *VAL2* and *VAL3*. (Default = -1.)

VAL2 -- Checking start time, where *VAL2* < *VAL3*. (Default = Start time of load step.)

VAL3 -- Checking end time, where *VAL3* > *VAL2*. (Default = End time of load step.)

VAL4 -- SOLID187 element type ID (defined prior to issuing this command). Valid only for SOLID185 or SOLID186 components in a NLAD-ETCHG analysis.

OFF

Disable the defined criteria for the specified component(s).

Criterion

Type of criterion to apply to the selected component(s):

CONTACT

Contact-based. Valid only for *Action* = ADD, *Action* = LIST, or *Action* = DELETE.

ENERGY

Energy-based. Valid only for *Action* = ADD, *Action* = LIST, or *Action* = DELETE.

BOX

A position-based criterion, defined by a box. Valid only for *Action* = ADD, *Action* = LIST, or *Action* = DELETE.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MESH

A mesh-quality-based criterion. Valid only for *Action* = ADD, *Action* = LIST, or *Action* = DELETE.

AM

Criterion used with Additive Manufacturing analyses. Valid only for *Action* = ADD, *Action* = LIST, or *Action* = DELETE.

ALL

All criteria and options. Valid only for *Action* = LIST or *Action* = DELETE. *Option* and all subsequent arguments are ignored.

Option

Criterion option to apply to the selected component(s):

NUMELEM

For target elements only, defines the minimum number of contact elements to contact with each target element. If this criterion is not satisfied, the program refines the contact elements and the associated solid elements. For this option, *VAL1* must be a positive integer.

Valid only for *Criterion* = CONTACT and *Action* = ADD, LIST, or DELETE.

MEAN

Checks the strain energy of any element that is part of the defined component for the condition $E_e \geq c_1 * E_{total} / NUME$ (where c_1 = *VAL1*, E_{total} is the total strain energy of the component, and *NUME* is the number of elements of the component). If this criterion is satisfied at an element, the program refines the element. *VAL1* must be non-negative. Default = 1.

Valid only for *Criterion* = ENERGY and *Action* = ADD, LIST, or DELETE.

XYZRANGE

Defines the location box in which all elements within are to be split or refined. Up to six values following the *Option* argument (representing the x1, x2, y1, y2, z1, and z2 coordinates) are allowed. An unspecified coordinate is not checked.

Valid only for *Criterion* = BOX and *Action* = ADD, LIST, or DELETE.

OCTREE

Sets octree options for AM octree adaptive meshing for additive simulations.

VAL1 – Number of layers to keep at fine mesh resolution between the current, top layer and the layers to be remeshed. Default = 2.

VAL2 – Number of buffer elements to keep at fine mesh resolution between the part edges and the remeshed elements. Default = 2.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid only for *Criterion* = AM and *Action* = ADD, LIST, or DELETE.

SKEWNESS

Mesh-quality-control threshold for elements SOLID187, SOLID285, and SOLID227:

VAL1 – Defines skewness. Valid values: 0.0 through 1.0. Default = 0.9.

VAL2 – Maximum Jacobian ratio at element integration points (SOLID187 and SOLID227 only). Valid values: 0.0 to 1.0. Default = 0.1.

Valid only for *Criterion* = MESH and *Action* = ADD, LIST, or DELETE.

SHAPE

Mesh-quality control threshold for elements PLANE182 and PLANE222. Also applies to SOLID185 and SOLID186 in a NLAD-ETCHG analysis.

VAL1 -- Maximum corner angle of an element in degrees. Valid values are 0 through 180. Default = 160 (2D analysis) or 155 (3D analysis). An element is remeshed when any of its corner angles reach the specified value.

Valid only for *Criterion* = MESH and *Action* = ADD, LIST, or DELETE.

WEAR

For contact elements having surface wear specified (**TB (p. 1899)**,WEAR) only, defines *VAL1* as a critical ratio of magnitude of wear to the average depth of the solid element underlying the contact element. Once this critical ratio is reached for any element, the program morphs the mesh to improve the quality of the elements. *VAL1* must be a positive integer.

Valid only for *Criterion* = CONTACT and *Action* = ADD, *Action* = LIST, or *Action* = DELETE. Cannot be combined with any other option during solution.

CZM

For contact elements with cohesive zone material (**TB (p. 1899)**,CZM) only, defines *VAL1* as a critical value of change in released energy due to debonding between reference and current substep, and *VAL2* as the critical value for the change in the damage parameter between neighboring elements. Both values can be applied separately or together.

When the critical value is reached (for either of the defined options) for one contact element, the solid elements underlying that contact element and the corresponding deformable target element are selected as candidates for remeshing.

Required: *VAL1* > 0, and $1 \leq$ *VAL2* ≤ 1. No default.

If issuing **NLMESH (p. 1240)**,,,*VAL3* where *VAL3* > 1.0, the criterion also checks for coarsening of solid elements underlying the debonded contact element and the corresponding deformable target element.

Valid only for *Criterion* = CONTACT and *Action* = ADD, LIST, or DELETE. Combining the CZM criterion with mesh-quality-based criteria may be necessary to improve distorted elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

All options. Valid only for *Action* = LIST or *Action* = DELETE. All subsequent arguments are ignored.

Notes

If a specified component (*Component*) is an assembly, the defined criterion applies to all element components included in the assembly.

All components must be defined and selected *before* the first solve (**SOLVE (p. 1822)**), although their nonlinear adaptivity criteria can be modified from load step to load step, and upon restart. For nonlinear adaptivity to work properly, ensure that all components are selected before each solve.

After issuing this command to define a new criterion, the new criterion becomes active. The program checks the new criterion once per load step, roughly in mid-loading (unless this behavior is changed via *Action* = ON).

When a criterion is defined, it overwrites a previously defined criterion (if one exists) through the same component, or through the component assembly that includes the specified component.

During solution, the same criteria defined for an element through different components are combined, and the tightest criteria and action control (*Action*,ON,,,*VAL1*) are used. If an ON action is defined by a positive *VAL1* value through one component and a negative *VAL1* value through another, the program uses the positive value.

When the AM octree option is specified (*Action* = ADD, *Criterion* = AM, *Option* = OCTREE), the checking frequency (*Action*,ON,,,*VAL1*), checking start time (*Action*,ON,,,*VAL2*), and checking end time (*Action*,ON,,,*VAL3*) control the checking layer frequency, start layer, and end layer respectively. If start and end layers are not specified, the start layer will default to the checking frequency, and the end layer will default to the final layer of the AM simulation.

Action = ON

If *VAL1* < 0, the program checks *VAL1* points between *VAL2* and *VAL3*. The time interval between each check points is determined by (*VAL3* - *VAL2*) / (*VAL1* + 1), with the first check point as close to *VAL2* + (*VAL3* - *VAL2*) / (*VAL1* + 1) as possible. Fewer check points can be used if the number of substeps during solution is insufficient (as the program can only check at the *end* of a substep).

If *VAL2* (start time) and/or *VAL3* (end time) are unspecified or invalid, the program uses the start and/or end time (respectively) of the load step.

Option = SKEWNESS

VAL1 applies to tetrahedral elements SOLID187, SOLID227, and SOLID285. When the skewness of an element is >= *VAL1*, the element is used as the core (seed) element of the remeshed region(s). The most desirable skewness value is 0, applicable when the element is a standard tetrahedral element; the highest value is 1, applicable when the element becomes flat with zero volume. To bypass skewness checking (not recommended), set *VAL1* = 0.

VAL2 represents the Jacobian ratio and is required for tetrahedral elements SOLID187 and SOLID227. When the maximum Jacobian ratio of an element is <= *VAL2*, the element is used as the core (seed)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

element of the remeshed region(s). The most desirable maximum Jacobian ratio is 1, when the element is a standard tetrahedral element; the lowest reported value is -1, when the element is turned inside out. To bypass maximum Jacobian ratio checking (not recommended), set *VAL2* = 0.

If this criterion is used with any other criteria defined for the same component, and a mesh change is requested at the same substep, *all* criteria defined are considered together. For more information about this special case, see Simultaneous Quality- and Refinement-Based Remeshing in the *Nonlinear Adaptivity Analysis Guide*.

Resources

For more information about skewness, maximum Jacobian ratio, and remeshing, see Nonlinear Mesh Adaptivity in the *Nonlinear Adaptivity Analysis Guide*.

For more granular control of the source mesh geometry, see **NLMESH (p. 1240)**.

Menu Paths

This command cannot be accessed from a menu.

NLDIAG, *Label*, *Key*, *MAXFILE*

Sets nonlinear diagnostics functionality.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Diagnostic function:

NRRE

Store the Newton-Raphson residuals information.

EFLG

Identify or display elements or nodes that violate the criteria.

CONT

Write contact information to a single `Jobname.cnd` diagnostic text file during solution.

Key

Diagnostic function characteristics:

OFF or 0

Suppresses writing of diagnostic information (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON or 1

Writes diagnostic information to the Jobname.nd*xxx*, Jobname.nr*xxx*, or Jobname.cnd file. (If *Label* = CONT, this option is the same as the SUBS option described below.)

ITER

Writes contact diagnostic information at each iteration. Valid only when *Label* = CONT.

SUBS

Writes contact diagnostic information at each substep. Valid only when *Label* = CONT.

LSTP

Writes contact diagnostic information at each load step. Valid only when *Label* = CONT.

STAT

Lists information about the diagnostic files in the current working directory.

DEL

Deletes all diagnostic files in the current working directory.

MAXFILE

Maximum number of diagnostic files to create. Valid values are 1 through 999. Default = 4. Valid only when *Label* = NRRE or EFLG.

Information is written to Jobname.nd*xxx* or Jobname.nr*xxx*, where *xxx* iterates from 001 through *MAXFILE*. When the specified maximum number of diagnostic files is reached, the counter resets to 001 and earlier files are overwritten. The *MAXFILE* value specified for this *Label* function applies until a new value is specified.

Command Default

No nonlinear diagnostic files are written.

Notes

The **NLDIAG (p. 1224)** command is a nonlinear diagnostics tool valid for nonlinear analyses that include structural degrees of freedom. It is a debugging tool for use when you must restart after an unconverged solution. The command creates Jobname.nd*xxx*, Jobname.nr*xxx*, or Jobname.cnd files in the working directory to store the information you specify.

For more information, see Performing Nonlinear Diagnostics.

Newton-Raphson Residual

Issue the **NLDIAG (p. 1224)**,NRRE,ON command to create Jobname.nr*xxx* diagnostic files (for each equilibrium iteration after the first) in which to store the relevant Newton-Raphson residual information

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of FX, FY, FZ (forces), MX, MY, MZ (moments), HEAT (heat flow), AMPS (current flow), CHRG (electric charge), or RATE (diffusion flow rate) for the last *MAXFILE* equilibrium iterations.

Issue a **NLDPOST (p. 1229)**,NRRE,STAT command to list the load step, substep, time, and equilibrium iteration corresponding to each of the Jobname.nr*xxx* diagnostic files in the working directory, then issue a **PLNSOL (p. 1425)**,NRRES,,,,*FileID* command to point to the file from which you want to create a contour plot of your Newton-Raphson residuals.

If you restart or issue a new **SOLVE (p. 1822)** command, any Jobname.nr*xxx* diagnostic files in the current (working) directory are overwritten.

Element Components That Violate Criteria

Issue a **NLDIAG (p. 1224)**,EFLG,ON command to create Jobname.nd*xxx* diagnostic files which store IDs for elements violating the following criteria:

- Too large a distortion (HDST)
- Elements contain nodes that have near zero pivots (PIVT) for nonlinear analyses
- Too large a plastic/creep (EPPL/EPCR) strain increment (**CUTCONTROL (p. 445)**)
- Elements for which mixed u-P constraints are not satisfied (mixed U-P option of 18*x* solid elements only) (MXUP)
- Hyperelastic element (EPHY), cohesive zone material (EPCZ), or damage strain (EPDM) not converged
- Radial displacement (RDSP) not converged
- MPC184 multipoint constraint elements using KEYOPT(1) = 6 through 16 with the Lagrange multiplier option fail to satisfy constraint conditions (184J)

For **NLDIAG (p. 1224)**,EFLG,ON, all Jobname.nd*xxx* diagnostic files (for each equilibrium iteration after the first) in the current (working) directory are deleted when you issue a new **SOLVE (p. 1822)** command (or restart).

In the solution processor (**/SOLU (p. 1821)**), use the STAT option to list the active status of this command. In the postprocessor (**/POST1 (p. 1461)**), issue a **NLDPOST (p. 1229)**,EFLG,STAT command to list the load step, substep, time, and equilibrium iteration corresponding to each of the Jobname.nd*xxx* diagnostic files in the working directory, then issue a **NLDPOST (p. 1229)**,EFLG,CM,*FileID* command to create element components that violate the criteria.

Contact Diagnostic

Issue the **NLDIAG (p. 1224)**,CONT,ON command to create a Jobname.cnd diagnostic file which stores contact information for all defined contact pairs at all substeps. Alternatively, you may issue one of the following commands to store contact information at a specific frequency:

- **NLDIAG (p. 1224)**,CONT,ITER to write at each iteration
- **NLDIAG (p. 1224)**,CONT,SUBS to write at each substep (default)
- **NLDIAG (p. 1224)**,CONT,LSTP to write at each load step

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contact diagnostic information is available for elements CONTA172, CONTA174, CONTA175, and CONTA177; it is not available for CONTA178.

Diagnostic file `Jobname.cnd` is written during solution and lists, on a pair-base, the following contact information:

- Contact pair ID[1] (p. 1228)
- Number of contact elements in contact[2] (p. 1228)
- Number of contact elements in "sticking" contact status
- Maximum chattering level
- Maximum contact penetration/Minimum gap[3] (p. 1228)
- Maximum geometric gap
- Maximum normal contact stiffness
- Minimum normal contact stiffness
- Maximum resulting pinball
- Maximum elastic slip distance
- Maximum tangential contact stiffness
- Minimum tangential contact stiffness
- Maximum sliding distance (algebraic sum)
- Maximum contact pressure
- Maximum friction stress
- Average contact depth
- Maximum geometric penetration
- Number of contact points having too much penetration
- Contacting area
- Maximum contact damping pressure
- Maximum tangential contact damping stress
- Maximum total sliding distance (GSLID), including near-field
- Minimum total sliding distance (GSLID), including near-field
- Maximum normal fluid penetration pressure on contact surface
- Maximum normal fluid penetration pressure on target surface

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Total volume lost due to wear for the contact pair
- Total strain energy due to contact constraint[6] (p. 1229)
- Total frictional dissipation energy[6] (p. 1229)
- Total contact stabilization energy[6] (p. 1229)
- Ansys Workbench contact pair ID[4] (p. 1228)
- Total force due to contact pressure - X component
- Total force due to contact pressure - Y component
- Total force due to contact pressure - Z component[5] (p. 1228)
- Total force due to tangential stress - X component
- Total force due to tangential stress - Y component
- Total force due to tangential stress - Z component[5] (p. 1228)
- Number of contact points having too much sliding for small sliding contact
- Pair-based force convergence norm[7] (p. 1229)
- Pair-based force convergence criterion[7] (p. 1229)
- Maximum tangential fluid penetration pressure on contact surface
- Maximum tangential fluid penetration pressure on target surface
- Maximum sliding distance for closed contact in the current substep

1. Contact pair ID. A positive number refers to a real constant ID for a pair-based contact definition. A negative number refers to a section ID of a surface in a general contact definition. (See Comparison of Pair-Based Contact and General Contact.)
2. Number of contact elements in contact. Other values are interpreted as follows:

   0 indicates that the contact pair is in near-field contact status.

   -1 indicates that the contact pair is in far-field contact status.

   -2 indicates that the contact pair is inactive (symmetric to asymmetric contact).
3. A positive value indicates penetration and a negative value indicates a gap. If the contact pair has a far-field contact status, penetration and gap are not available and the value stored is the current pinball radius.
4. Intended primarily for internal use in the contact tracking of Ansys Workbench.
5. In a 3D model, the reported item is total force along the Z-axis. In a 2D axisymmetric model (with or without ROTY), the reported item is maximum torque that can potentially act on the Y-axis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. The pair-based dissipation energy and stabilization energy do not include contributions from contact elements that are in far-field. The pair-based strain energy does not include the frictional dissipation energy and stabilization energy; it only contains an elastic recovery energy when the contact status changes from closed to open.

7. The program uses a default tolerance value of 0.1 to calculate the *pair-based force convergence norm* and *pair-based force convergence criterion*. This is not a check for local convergence. It is for monitoring purposes only and is useful for nonlinear contact diagnostics.

In the solution processor (**/SOLU (p. 1821)**), use the **NLDIAG (p. 1224)**,CONT,STAT command to list the active status of the contact information. If you subsequently issue a new **SOLVE (p. 1822)** command (or restart), the `Jobname.cnd` diagnostic file in the current (working) directory is not deleted; information is appended to it. Delete the existing diagnostic file (**NLDIAG (p. 1224)**,CONT,DEL command) if you do not want to retain diagnostic information from previous solutions.

Menu Paths

Main Menu>Solution>Diagnostics>Nonlinear Diagnostics

NLDPOST, *Label*, *Key*, *FileID*, *Prefix*

Gets element component information from nonlinear diagnostic files.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Specifies the type of command operation:

EFLG

Element flag for nonlinear diagnostics.

NRRE

Newton-Raphson residuals.

Key

Specifies the command action:

STAT

List information about the diagnostic files (`Jobname.nd`*xxx* or `Jobname.nr`*xxx*) in the current directory.

For *Label* = EFLG, the listing gives a summary that associates the loadstep, substep, time, equilibrium iteration number, cumulative iteration number, and the number of elements that fail each criteria with a specific file ID (`Jobname.nd`*xxx*). Use the list to create element components (via the CM option) based on the cumulative iteration number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For *Label* = NRRE, the listing provides a summary that associates the loadstep, substep, time, equilibrium iteration number, and cumulative iteration number with a specific file ID (Jobname.nr*xxx*). Use the list to identify the respective file ID for creating Newton-Raphson residual contour plots (**PLNSOL (p. 1425)**,NRRE,...,*FileID*).

DEL

Delete Jobname.nd*xxx* or Jobname.nr*xxx* files in the working directory, if any exist.

CM

Create components for elements that violate criteria. This value is valid only when *Label* = EFLG.

FileID

Valid only when *Label* = EFLG and *Key* = CM, this value specifies file IDs:

IDnum

The file ID number. Creates the element components from the diagnostic files corresponding to the specified file ID number in the working directory.

ALL

Creates element components from all available diagnostic files residing in the working directory. This value is the default if you do not specify an *IDnum* value.

Prefix

Sets the prefix name for components. Specify up to 21 alphanumeric characters.

Command Default

The **NLDPOST (p. 1229)** command returns no nonlinear diagnostic information.

Notes

Based on the nonlinear diagnostic results (created via the **NLDIAG (p. 1224)**,EFLG command), the **NLDPOST (p. 1229)** command creates element components with predefined names.

The following table lists the diagnostic criteria and component names (with specified prefix and without). Here *xxx* corresponds to the file ID (*FileID*) of Jobname.nd*xxx* or Jobname.nr*xxx*.

Elements with:	If prefix is specified:	Without prefix specified:
Too large a plastic strain increment	*prefix_xxx*_eppl	nd*xxx*_eppl
Too large a creep strain increment	*prefix_xxx*_epcr	nd*xxx*_epcr nd*xxx*_ephy

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Elements with:	If prefix is specified:	Without prefix specified:
		ndxxx_epcz ndxxx_epdm
Too much distortion	*prefix_xxx*_hdst	ndxxx_hdst ndxxx_rdsp
Unsatisfied mixed u-P constraints	*prefix_xxx*_mxup	ndxxx_mxup
Unconverged MPC184 multipoint constraint elements	*prefix_xxx*_184j	ndxxx_184j
Negative/small pivots	*prefix_xxx*_pivt	ndxxx_pivt

If you have trouble viewing specific element components, see Viewing Hidden Element Components in the *Basic Analysis Guide*.

For more information, see Performing Nonlinear Diagnostics.

Menu Paths

Main Menu>General Postproc>Nonlinear Diagnostics

NLGEOM, *Key*

Includes large-deflection effects in a static or full transient analysis.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Large-deflection key:

OFF

Ignores large-deflection effects (that is, a small-deflection analysis is specified). This option is the default.

ON

Includes large-deflection (large rotation) effects or large strain effects, according to the element type.

Command Default

Large-deflection effects are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Large-deflection effects are categorized as either large deflection (or large rotation) or large strain, depending on the element type. These are listed (if available) under Special Features in the input data table for each element in the *Element Reference*. When large deflection effects are included (**NLGEOM (p. 1231)**,ON), stress stiffening effects are also included automatically.

If used during the solution (**/SOLU (p. 1821)**), this command is valid only within the first load step.

In a large-deflection analysis, pressure loads behave differently than other load types. For more information, see Load Direction in a Large-Deflection Analysis.

The gyroscopic matrix (that occurs due to rotational angular velocity) does not support large-deflection effects. The theoretical formulations for the gyroscopic matrix support small deflection (linear formulation) only.

When large-deflection effects are included in a substructure or CMS transient analysis use pass, the **OUTRES (p. 1336)** command ignores *DSUBres* = ALL.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

NLHIST,*Key*,*Name*,*Item*,*Comp*,*NODE*,*ELEM*,*SHELL*,*LAYER*,*STOP_VALUE*,*STOP_COND*

Specify results items to track during solution.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Specifies the command operation:

NSOL

Nodal solution data.

ESOL

Element nodal data.

PAIR

Contact data (for pair-based contact).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GCN

Contact data (for general contact).

RSEC

Result section data.

STAT

Displays a list of items to track.

OFF or 0

Deactivates tracking of all variables. This value is the default.

ON or 1

Activates tracking of all variables. Tracking also activates whenever any specification changes.

DEL

Removes the specified variable from the set of result items to track. If *Name* = ALL (default), all specifications are removed.

Name

The 32-character user-specified name.

Item,Comp

Predetermined output item and component label for valid elements. See the *Element Reference* for more information.

NODE

Number identifying one of the following:

Valid node number (if *Key* = NSOL or ESOL).

Valid real constant set number identifying a contact pair (if *Key* = PAIR).

Valid section ID number identifying a surface of a general contact definition (if *Key* = GCN).

Valid real constant set number identifying a result section (if *Key* = RSEC).

NODE is required input when *Key* = NSOL, ESOL, PAIR, GCN, or RSEC.

ELEM

Valid element number for element results. Used for ESOL items. If *ELEM* is specified, then a node number that belongs to the element must also be specified in the *NODE* field.

SHELL

Valid labels are TOP, MID or BOT. This field can specify the location on shell elements for which to retrieve data. Used only for element nodal data (ESOL).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LAYER

Layer number (for layered elements only). Used only for element nodal data (ESOL).

STOP_VALUE

Critical value of the tracked variable. This value is used to determine if the analysis should be terminated.

STOP_COND

Specifies the conditional relationship between the variable being tracked and the *STOP_VALUE* upon which the analysis will be terminated:

-1

Terminate the analysis when the tracked variable is less than or equal to *STOP_VALUE*.

0

Terminate the analysis when the tracked variable equals *STOP_VALUE*.

1

Terminate the analysis when the tracked variable is greater than or equal to *STOP_VALUE*.

Notes

The **NLHIST (p. 1232)** command is a nonlinear diagnostics tool that enables you to monitor diagnostics results of interest in real time during a solution.

You can track a maximum of 50 variables during solution. The specified result quantities are written to the file `Jobname.nlh`. Nodal results and contact results are written for every converged substep (irrespective of the **OUTRES (p. 1336)** command setting), while element results are written only at time points specified via the **OUTRES (p. 1336)** command. Result section data is written only at time points specified via the **OUTPR (p. 1332)**,RSO,*Freq* command.

For time points where element results data is not available, a very small number is written instead. If the conditions for contact to be established are not satisfied, 0.0 will be written for contact results.

Results tracking is available for:

- nonlinear structural analyses (static or transient)
- nonlinear steady-state thermal analyses
- transient thermal analyses (linear or nonlinear)
- nonlinear coupled structural-thermal analyses (static or transient)

All results are tracked in the Solution Coordinate System (that is, nodal results are in the nodal coordinate system and element results are in the element coordinate system).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contact results can be tracked for elements CONTA172, CONTA174, CONTA175, and CONTA177; they cannot be tracked for CONTA178.

Result section data can be tracked for elements CONTA172 and CONTA174.

When contact results (*Key* = PAIR or GCN) or result section data (*Key* = RSEC) are tracked , the user-specified name (*Name* argument) is used to create a user-defined parameter. This enables you to monitor the parameter during solution. As an example, you can use a named parameter to easily convert the contact stiffness units from FORCE/LENGTH^3 to FORCE/LENGTH based on the initial contact area CAREA. Be sure to specify *Name* using the APDL parameter naming convention.

The *STOP_VALUE* and *STOP_COND* arguments enable you to automatically terminate the analysis when a desired value for a tracked contact result or section result has been reached.

The `Jobname.nlh` file is an ASCII file that lists each time point at which a converged solution occurs along with the values of the relevant result quantities.

The GUI option **Solution> Results tracking** provides an interface to define the result items to be tracked. The GUI also allows you to graph one or more variables against time or against other variables during solution. You can use the interface to graph or list variables from any `.nlh` file generated by Mechanical APDL.

You can also track results during batch runs. Either access the launcher and select **Run Results Tracker Utility** from the **Tools** menu, or type `nlhist241` at the command line. Use the supplied file browser to navigate to your `Jobname.nlh` file, and click on it to invoke the tracking utility. You can use this utility to read the file at any time, even after the solution is complete (the data in the file must be formatted correctly).

Table 194: NLHIST - Valid NSOL (p. 1281) Item and Component Labels

Item	Comp	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
F	X, Y, Z	X, Y, or Z structural reaction force.
M	X, Y, Z	X, Y, or Z structural reaction moment.
TEMP [1]	-	Temperature.
TEMP	MAX, MIN	Maximum or minimum temperature in the model.
HEAT [2]	-	Reaction heat flow.

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP.

2. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . ., HTOP instead of HEAT.

Table 195: NLHIST - Valid ESOL (p. 672) Item and Component Labels

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
"	INT	Stress intensity.
"	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT	Elastic strain intensity.
"	EQV	Elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT	Plastic strain intensity.
"	EQV	Plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT	Creep strain intensity.
"	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT	Thermal strain intensity.
"	EQV	Thermal equivalent strain.
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain.
"	1, 2, 3	Principal diffusion strain.
"	INT	Diffusion strain intensity.
"	EQV	Diffusion equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.

ETABLE items are not supported for **ESOL (p. 672)** items.

PAIR solution quantities are output on a "per contact pair" basis. GCN solution quantities are output on a “per general contact section” basis. (See Comparison of Pair-Based Contact and General Contact.) As

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a consequence, the corresponding values listed in the `Jobname.nlh` file represent a minimum or a maximum over the associated contact pair or general contact surface, as detailed in the table below.

Table 196: NLHIST - Valid Contact (PAIR or GCN) Item and Component Labels

Item	Comp	Description
CONT	ELCN	If >0, number of contact elements in contact. Other values are interpreted as follows: 0 indicates the contact pair (or GCN surface) is in near-field contact status. -1 indicates the contact pair (or GCN surface) is in far-field contact status. -2 indicates that the contact pair (or GCN surface) is inactive (symmetric to asymmetric contact).
"	ELST	Number of contact elements in sticking contact status
"	CNOS	Maximum chattering level
"	PENE	Maximum penetration (or minimum gap) [1]
"	CLGP	Maximum geometric gap
"	SLID	Maximum total sliding distance (algebraic sum)
"	SLMX	Maximum total sliding distance for closed contact in the current substep
"	ESLI	Maximum elastic slip distance
"	KNMX	Maximum normal contact stiffness
"	KTMX	Maximum tangential contact stiffness
"	KNMN	Minimum normal contact stiffness
"	KTMN	Minimum tangential contact stiffness
"	PINB	Maximum pinball radius
"	PRES	Maximum contact pressure
"	SFRI	Maximum frictional stress
"	CNDP	Average contact depth
"	CLPE	Maximum geometric penetration
"	LGPE	Number of contact points having too much penetration
"	CAREA	Contacting area
"	NDMP	Maximum contact damping pressure
"	TDMP	Maximum tangential contact damping stress
"	GSMX	Maximum total sliding distance (GSLID), including near-field
"	GSMN	Minimum total sliding distance (GSLID), including near-field
"	FPSC	Maximum normal fluid penetration pressure on contact surface
"	FPST	Maximum normal fluid penetration pressure on target surface
"	WEAR	Total volume lost due to wear for the contact pair (not available for general contact, *Key* = GCN)
"	CTEN	Total strain energy due to contact constraint [2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
"	CFEN	Total frictional dissipation energy [2]
"	CDEN	Total contact stabilization energy [2]
"	CFNX	Total force due to contact pressure - X component [4]
"	CFNY	Total force due to contact pressure - Y component [4]
"	CFNZ	Total force due to contact pressure - Z component [4] [5]
"	CFSX	Total force due to tangential stress - X component [4]
"	CFSY	Total force due to tangential stress - Y component [4]
"	CFSZ	Total force due to tangential stress - Z component [4] [5]
"	CTRQ	Maximum torque in an axisymmetric analysis with MU = 1.0
"	LGSL	Number of contact points having too much sliding for small sliding contact
"	NORM	Pair-based force convergence norm [3]
"	CRIT	Pair-based force convergence criterion [3]
"	FPTC	Maximum tangential fluid penetration pressure on contact surface
"	FPTT	Maximum tangential fluid penetration pressure on target surface

1. For PENE, a positive value indicates a penetration, and a negative value indicates a gap. If the contact pair (or GCN surface) has a far-field contact status, penetration and gap are not available, and the value stored for PENE is the current pinball radius.

2. The pair-based dissipation energy (CFEN) and stabilization energy (CDEN) do not include contributions from contact elements that are in far-field. The pair-based strain energy (CTEN) does not include the frictional dissipation energy and stabilization energy; it only contains an elastic recovery energy when the contact status changes from closed to open.

3. The program uses a default tolerance value of 0.1 to calculate the *pair-based force convergence norm* and *pair-based force convergence criterion*. This is not a check for local convergence. It is for monitoring purposes only and is useful for nonlinear contact diagnostics.

4. If the specified contact pair is a rigid surface or force-distributed constraint that includes stress stiffening effects, this quantity represents a total constraint force or moment at the pilot node as shown below:

CFNX	Total constraint force - X component
CFNY	Total constraint force - Y component
CFNZ	Total constraint force - Z component
CFSX	Total constraint moment - X component
CFSY	Total constraint moment - Y component
CFSZ	Total constraint moment - Z component

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. For the case of 2D axisymmetric with torsion (CONTA172 with a ROTY DOF), CFNZ and CFSZ represent moments along the Y direction.

Table 197: NLHIST - Valid Result Section (RSEC) Item and Component Labels

Item	Comp	Description
REST	SECF	Total section force
"	SECM	Total section moment
"	AXST	Section axial stress
"	BDST	Section bending stress
"	SPTX	Section center X coordinate
"	SPTY	Section center Y coordinate
"	SPTZ	Section center Z coordinate
"	THXY	Rotation about local z
"	THYZ	Rotation about local x
"	THZX	Rotation about local y

Menu Paths

Main Menu>Solution>Results Tracking

NLIST,*NODE1*,*NODE2*,*NINC*,*Lcoord*,*SORT1*,*SORT2*,*SORT3*,*KINTERNAL*

Lists nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2*, *NINC

List nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL (default), *NODE2* and *NINC* are ignored and all selected nodes (**NSEL (p. 1266)**) are listed. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Lcoord

Coordinate listing key:

(blank)

List all nodal information

COORD

Suppress all but the XYZ coordinates (shown to a higher degree of accuracy than when displayed with all information).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SORT1

First item on which to sort. Valid item names are NODE, X, Y, Z, THXY, THYZ, THXZ

SORT2, SORT3

Second and third items on which to sort. Valid item names are the same as for *SORT1*.

KINTERNAL

Internal nodes listing key:

(blank)

List only external nodes.

INTERNAL

List all nodes, including internal nodes.

Notes

Lists nodes in the active display coordinate system (**DSYS (p. 582)**). Nodal coordinate rotation angles are also listed (relative to the global Cartesian coordinate system).

Node listing can be in a sorted order (ascending). *SORT2*, for example, will be carried out on nodes having equal values of *SORT1*.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes

Utility Menu>List>Nodes

NLMESH, *Control*, *VAL1*, *VAL2*, *VAL3*

Controls remeshing in nonlinear adaptivity.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Control

The mesh-quality control to adjust:

NLAY

The *sculpting layer* adjustment:

VAL1 – The number of sculpting layers, beginning with detected seed elements. Valid for 2D and 3D analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default = 10 for 2D analysis, 2 for 3D analysis.

For local (partial) remeshing, this option helps the remesher to detect remeshing regions from the whole deformed model.

Generally, a larger *VAL1* leads to larger remeshing regions and tends to unite isolated multiple regions. A larger value also tends to result in better remeshing quality (and increases mapping and solution overhead accordingly).

Only elements with the same element and material properties as seed elements are included into the remeshing regions.

VAL1 = 0 is not valid, as the remeshing regions would contain only the detected seed elements, resulting in many small cavities within remeshing regions (especially if the specified mesh-quality metric threshold (**NLADAPTIVE (p. 1219)**) is relatively large).

VAL2 – Same as *VAL1*, except that *VAL1* controls remeshing to remove distortion and *VAL2* controls element refinement. Default = 1 for 2D analysis, 2 for 3D analysis.

Not used in a NLAD-ETCHG analysis.

For more information about this control, see Sculpting Layers Control in the *Nonlinear Adaptivity Analysis Guide*.

BDRA

The *boundary angle threshold* in degrees. Use this adjustment to retain geometry features of the original (source) mesh. Valid for both 2D and 3D analysis.

In a 3D analysis, this value is the dihedral angle (the angle between the normal vectors from two neighboring surface facets). In a 2D analysis, this value is the 2D patch boundary edge normal angle. If the edge angle or dihedral angle is larger than the specified threshold, the node shared by 2D edges or edges shared by 3D facets are retained during remeshing.

Valid values: 0 < *VAL1* < 80.

Default = 10 for 2D analysis, 15 for 3D analysis.

Generally, a larger *VAL1* improves the quality of the new mesh (and may even repair local tiny edges or facets of poor quality). Too large a value, however, may also smooth out some geometric features, leading to slightly different results and possibly causing convergence issues in the substeps immediately following remeshing.

For more information about this control, see Boundary-Angle and Edge-Angle Control in the *Nonlinear Adaptivity Analysis Guide*.

AEDG

The *edge angle threshold* in degrees. Valid for 3D analysis only.

Use this control to split 3D patch segments. The edge angle is the angle between adjacent surface segment edges sharing a node. If the edge angle is larger than the specified threshold (*VAL1*), the segment splits and the node is automatically treated as a node to be retained.

Default = 10.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Generally, a larger *VAL1* improves the quality of the new mesh, but may result in loss of feature nodes. The effect is similar to that of boundary angles (*Control* = BDRA).

For more information about this control, see Boundary-Angle and Edge-Angle Control in the *Nonlinear Adaptivity Analysis Guide*.

SRAT

The *global sizing ratio*. Valid for 2D and 3D analysis.

VAL1 – Global sizing ratio for remeshing.

This value controls remeshing to remove mesh distortion. Default = 1.0. The default value results in the new mesh having a size similar to that of the original mesh.

Generally, set *VAL1* $\geq$ 0.7. The model can be refined (< 1.0) or coarsened (> 1.0) up to 3x depending on the mesh-sizing gradient and number of 3D elements, and approximately 2x for 2D elements.

VAL2 – Similar to *VAL1*, but controls element refinement. Default = 0.75.

VAL3 – Equivalent element-size expansion ratio.

Used only for contact elements with cohesive zone material (**NLADAPTIVE (p. 1219)**,,,CONTACT,CZM).

VAL3 > 1.0 activates mesh coarsening in regions of debonding.

Recommendation: For most common mode I or mode II delamination problems, 1.5 < *VAL3* < 2 for 2D and 3D analysis. For complex models (such as mixed-mode, nonplanar delamination surfaces, and models with multiple surfaces close to each other), you must determine the most appropriate *VAL3*. Remeshing frequency and the quality of the starting mesh also affect your choice of *VAL3*.

VAL3 has no default value.

Not used in an NLAD-ETCHG analysis.

For more information about this control, see Global Sizing Control in the *Nonlinear Adaptivity Analysis Guide*.

GRAD

Adjusts the *new mesh-sizing gradient* during remeshing. Valid for 2D and 3D analysis.

Valid values: *VAL1* = 0, 1, 2, or 3. Default = 2 for 2D analysis, 3 for 3D analysis.

VAL1 = 0 -- The mesh-sizing gradient is not retained. The new mesh is uniform and has an approximate average size on the entire remeshed domain(s), even if the original mesh has sizing gradients.

VAL1 = 1 -- The new mesh follows the original mesh-sizing gradient to retain the element averaged-edge length. This value tends to coarsen the mesh in the location of the distorted elements during remeshing.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL1 = 2 -- The new mesh follows the sizing gradient of the original mesh, with additional sizing compensation based on the element size change due to deformation during solution. This value tends to refine the mesh at the location of the distorted elements, where the distortion may have originated from deformation during solution.

VAL1 = 3 -- Similar to *VAL1* = 2, but assumes that perfect mesh quality is not required, thus avoiding over-refinement of minor distorted regions. Valid for 3D analysis only.

For more information about this control, see Gradient Control in the *Nonlinear Adaptivity Analysis Guide*.

QTOL

The new *mesh-acceptance tolerance* (PLANE182, PLANE222, SOLID187, SOLID227, and SOLID285).

VAL1 – Controls remeshing to remove distortion. Default = 0.05.

VAL2 – Controls element refinement. Default = 0.5.

For PLANE182 and PLANE222, *VAL2* is the only valid option (for mesh refinement), and the new mesh is accepted when (*MaxCornerAngleNew* - *MaxCornerAngleOld*) / *MaxCornerAngleOld* <= *QTOL*.

For SOLID285, the new mesh is accepted when (*SkewnessNew* - *SkewnessOld*) / *SkewnessOld* <= *QTOL*.

For SOLID187 and SOLID227, the new mesh is accepted when, in addition to skewness, (*JacobianOld* - *JacobianNew*) / *JacobianOld* <= *QTOL*.

The program uses both tolerance and mesh-quality parameters to determine whether or not a new mesh is accepted.

Not used in a NLAD-ETCHG analysis.

REFA

The *refinement algorithm* adjustment (PLANE182, PLANE222, and SOLID285). Valid for 2D and 3D analysis.

VAL1 –

SPLIT – Use mesh splitting instead of general remeshing. This is the only valid value.

If not specified, mesh refinement occurs via general remeshing (except for PLANE183).

Not used in a NLAD-ETCHG analysis.

TCOR

Coordinate truncation adjustments for nodal locations of the meshes during remeshing. Valid for 2D and 3D analysis.

VAL1 –

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON – Truncates the decimal value after the seventh position. Default behavior for PLANE182, PLANE222, and SOLID285 (augmented Lagrange and penalty contact formulations only).

OFF – No truncation occurs on the decimal value. Default behavior for SOLID187 and SOLID227 (all contact formulations), and PLANE182 and PLANE222 (Lagrange multiplier contact formulation only).

AGGR

Aggressive remeshing. Creates meshes with improved shape metrics. May change some global remeshing control parameters applied by other **NLMESH (p. 1240)** commands, and may increase remeshing time. Valid for both 2D and 3D nonlinear adaptivity analysis.

VAL1 –

ON – Enable aggressive remeshing.

OFF – Disable aggressive remeshing (default).

ELSZ

Reduces the set of remeshable (seed) elements used for remeshing by filtering out elements based on their size, preventing over- or under-refinement in the remeshing regions. Valid only in general remeshing using an energy-, position-, or contact-based refinement criterion.

Val1 is a user-defined lower bound of the element size, and *Val2* is a user-defined upper bound of the element size. If both are specified, seed elements selected via the specified criterion are filtered out if their size is <*Val1* or >*Val2*. If *Val1* is unspecified, only the size check with *Val2* occurs. If *Val2* is unspecified, only the size check with *Val1* occurs.

NSTR

Reduces the set of remeshable (seed) elements used for remeshing by filtering out elements based on their (maximum) equivalent stress level, preventing over- or under-refinement in the remeshing regions based on the stress state. Valid only in general remeshing using an energy-, position-, or contact-based refinement criterion.

Val1 is a user-defined lower bound of element equivalent stress, and *Val2* is a user-defined upper bound of the element equivalent stress. If both are specified, seed elements selected by the specified criterion are filtered out if their (maximum) equivalent stress is <*Val1* or >*Val2*. If *Val1* is unspecified, only the equivalent stress check with *Val2* occurs. If *Val2* is unspecified, only the equivalent stress check with *Val1* occurs.

Important: Before specifying this option, copy the integration-point results to the nodes (**ERESX (p. 658)**,NO).

NSTN

Reduces the set of remeshable (seed) elements used for remeshing by filtering out elements based on their (maximum) equivalent strain level, preventing over- or under-refinement in the remeshing regions based on the strain state. Valid only in general remeshing using an energy-, position-, or contact-based refinement criterion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val1 is a user-defined lower bound of the element equivalent strain, and *Val2* is a user-defined upper bound of the element equivalent strain. If both are specified, seed elements selected by the specified criterion are filtered out if their (maximum) equivalent strain is <*Val1* or >*Val2*. If *Val1* is unspecified, only the equivalent strain check with *Val2* occurs. If *Val2* is unspecified then only the equivalent strain check with *Val1* occurs.

Important: Before specifying this option, copy the integration-point results to the nodes (**ERESX (p. 658)**,NO).

LIST

Lists all mesh-quality control parameters.

If *VAL1* has been specified for a given mesh control, the most recently specified value is listed. If a value was not explicitly specified, the default value is listed (assuming that the problem has been solved at least once).

VAL1

Numerical input value that varies according to the specified *Control* option.

Valid for all *Control* options. Can be used when controlling remeshing for both distortion removal and for element refinement.

VAL2

Numerical input value that varies according to the specified *Control* option.

Valid only for these *Control* options: NLAY, SRAT, and QTOL. Also used for controlling remeshing for element refinement.

Notes

NLMESH (p. 1240) is a global control command enabling mesh-quality adjustments for remeshing in nonlinear adaptivity. The command can be used when components are associated with mesh-quality criteria (**NLADAPTIVE (p. 1219)** with *Criterion* = MESH, or another criterion with mesh change through general refinement).

Issue the **NLMESH (p. 1240)** command only in cases where advanced mesh-quality control is desirable for remeshing in nonlinear adaptivity. The control values specified apply to all components having mesh-quality-based criteria defined, or components having mesh change through general refinement, and can be modified at every load step during the nonlinear adaptive solution or when performing a restart analysis.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NLOG, *IR*, *IA*, - -, - -, *Name*, - -, - -, *FACTA*, *FACTB*

Forms the natural log of a variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

--,--

Unused fields.

Name

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--,--

Unused fields.

FACTA

Scaling factor applied to variable *IA* (defaults to 1.0).

FACTB

Scaling factor (positive or negative) applied to the operation (defaults to 1.0).

Notes

Forms the natural log of a variable according to the operation:

$$IR = FACTB * \mathrm{LN}(FACTA \times IA)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Natural Log

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NLOPT

Specifies "Nonlinear analysis options" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Nonlinear Options

NMODIF, *NODE*, *X*, *Y*, *Z*, *THXY*, *THYZ*, *THZX*

Modifies an existing node.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Modify coordinates of this node. If ALL, modify coordinates of all selected nodes (**NSEL (p. 1266)**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE*.

X, Y, Z

Replace the previous coordinate values assigned to this node with these corresponding coordinate values. Values are interpreted in the active coordinate system (R, θ, Z for cylindrical; R, θ, Φ for spherical or toroidal). Leaving any of these fields blank retains the previous value(s).

THXY

First rotation of nodal coordinate system about nodal Z (positive X toward Y). Leaving this field blank retains the previous value.

THYZ

Second rotation of nodal coordinate system about nodal X (positive Y toward Z). Leaving this field blank retains the previous value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

THZX

Third rotation of nodal coordinate system about nodal Y (positive Z toward X). Leaving this field blank retains the previous value.

Notes

Modifies an existing node. Nodal coordinate system rotation angles are in degrees and redefine any existing rotation angles. Nodes can also be redefined with the **N (p. 1207)** command.

See the **NROTAT (p. 1264)**, **NANG (p. 1208)**, and **NORA (p. 1252)** commands for other rotation options.

This command is also valid in the **/MAP (p. 1107)** processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>By Angles

Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Set of Nodes

Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Single Node

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>By Angles

---

NODES

Specifies "Nodes" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/NOERASE

Prevents the screen erase between displays.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Command Default

Issue the command with no arguments to prevent automatic screen erase before creating the next display.

Notes

Preventing the normal screen erase between requested displays allows you to overlay multiple views.

Clearing the screen with the **ERASE (p. 652)** command (**Utility Menu> PlotCtrls> Erase Options> Erase screen**) active simply clears the display area. Subsequent replots will provide the cumulative plots previously generated by the **/NOERASE (p. 1249)** command.

For 3D devices, you can issue **/DV3D (p. 584)**,DELS to suppress repeated screen overlays and generate clear contour plots.

Use the **/ERASE (p. 653)** command to reactivate automatic screen erase.

For 3D devices (**/SHOW (p. 1785)**,3D), the model in all active windows will be the same, even if you issue a different display command (**NPLOT (p. 1254)**, **EPLOT (p. 648)**, etc.) for each active window. Use the Multi-Plot command (**GPLOT (p. 866)**) to display different entities, in different windows, on 3D devices.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Erase Options>Erase between Plots

/NOLIST

Suppresses the data input listing.

SESSION (p. 11): List Controls (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Command Default

Issue the command with no arguments to suppress the data input listing.

Notes

Printout is suppressed until a **/GOLIST (p. 865)** command is read or the end of the listing is encountered.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is valid in any processor, but only within a batch run (**/BATCH (p. 217)**).

Menu Paths

This command cannot be accessed from a menu.

NOOFFSET, *Label*

Prevents the CDREAD (p. 291) command from offsetting specified data items

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Specifies items not to be offset.

NODE

Node numbers

ELEM

Element numbers

KP

Keypoint numbers

LINE

Line numbers

AREA

Area numbers

VOLU

Volume numbers

MAT

Material numbers

TYPE

Element type numbers

REAL

Real constant numbers

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CSYS

Coordinate system numbers

SECN

Section numbers

CP

Coupled set numbers

CE

Constraint equation numbers

CLEAR

All items will be offset

STATUS

Shows which items are specified *not*to be offset.

Notes

The **NOOFFSET (p. 1250)** command specifies data items not to be offset by a set of data read from a **CDREAD (p. 291)** command.

Menu Paths

This command cannot be accessed from a menu.

/NOPR

Suppresses the expanded interpreted input data listing.

SESSION (p. 11): List Controls (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Command Default

Issuing this command with no arguments suppresses the interpreted data input print out.

Notes

Suppresses printout of interpreted input data, including information labeled as "Notes." When this printout is not suppressed, the data input to the analysis is echoed to the output file in an expanded format. Printout is suppressed until a **/GOPR (p. 865)** or **/GO (p. 865)** command is read.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use of **/NOPR (p. 1251)** is not recommended when the graphical user interface (GUI) is active. The GUI sometimes issues "hidden" **/NOPR (p. 1251)** and **/GOPR (p. 865)** command sequences, which will countermand user-issued **/NOPR (p. 1251)** commands, thus making the use of **/NOPR (p. 1251)** in the GUI environment unpredictable.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

NORA, *AREA*, *NDIR*

Rotates nodal coordinate systems to surface normal

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

The area number containing the nodes to be rotated to their normals. If ALL, applies to all selected areas (see the **ASEL (p. 192)** command). If AREA = P, graphical picking is enabled.

NDIR

Direction of the normal. If NDIR = -1, the nodal coordinate system is rotated in the opposite direction of the surface normal. The default is the same direction as the surface normal.

Notes

The NORA command rotates the X-axis of the nodal coordinate system to the surface normal. The rotated nodal coordinate systems may be displayed through the **/PSYMB (p. 1552)** command. In case multiple areas are selected, there could be conflicts at the boundaries. If a node belongs to two areas that have a different normal, its nodal coordinate system will be rotated to the area normal with the lowest number. You can use the **AREVERSE (p. 181)** and **ANORM (p. 147)** commands to rotate the surface normals in the appropriate direction. Keep the following in mind when using the NORA command:

- If the nodal coordinate system is parallel to the global Cartesian system, it is not displayed through the **/PSYMB (p. 1552)** command.
- Previously specified rotation on the selected nodes are overridden.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NORL, *LINE*, *AREA*, *NDIR*

Rotates nodal coordinate systems perpendicular to line normal

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

Line number containing the nodes to be rotated. If ALL, applies to all selected lines (see the **LSEL (p. 1074)** command). If *LINE* = P, graphical picking is enabled.

AREA

The area number containing the selected lines. The normal of the line(s) selected is supposed to lie on this area. Defaults to the lowest numbered selected area containing the line number.

NDIR

Direction of the normal. If *NDIR* = -1, the nodal coordinate system is rotated in the opposite direction of the line normal. The default is the same direction as the surface normal.

Notes

The NORL command rotates the X-axis of the nodal coordinate perpendicular to the line normal. The rotated nodal coordinate systems may be displayed through the **/PSYMB (p. 1552)** command. In case multiple lines are selected, there could be conflicts at the boundaries. If a node belongs to two lines that have a different normal, its nodal coordinate system will be rotated to the line normal with the lowest number. Keep the following in mind when using the NORL command:

- If the nodal coordinate system is parallel to the global Cartesian system, it is not displayed through the **/PSYMB (p. 1552)** command.
- Previously specified rotation on the selected nodes are overridden.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area

/NORMAL, *WN*, *KEY*

Allows displaying area elements by top or bottom faces.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEY

Display key:

0

No face distinction.

1

Show only area elements having their positive normals directed toward the viewing point.

-1

Show only area elements having their positive normals directed away from the viewing point.

Command Default

Do not distinguish between top face and bottom face elements.

Notes

/NORMAL (p. 1253) allows you to select area elements and area plots by the top or bottom faces. It is useful for checking the normal directions on shell elements. The positive normal (element Z direction) is defined by the right-hand rule following the node I, J, K, L input direction. This command is available only with raster or hidden-line displays, for WIN32 or X11 2D displays only.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Shell Normals

NPLOT, *KNUM*

Displays nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KNUM

Node number key:

0

No node numbers on display.

1

Include node numbers on display. See also **/PNUM (p. 1457)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Produces a node display. Only selected nodes (**NSEL (p. 1266)**) are displayed. Elements need not be defined. See the **DSYS (p. 582)** command for display coordinate system.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area

Utility Menu>Plot>Nodes

NPRINT, *N*

Defines which time points stored are to be listed.

POST26 (p. 58): Listing (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

N

List data associated with every *N* time (or frequency) point(s), beginning with the first point stored (defaults to 1).

Command Default

List all stored points.

Notes

Defines which time (or frequency) points within the range stored are to be listed.

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NREAD, *Fname*, *Ext*, - -

Reads nodes from a file.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to NODE if *Fname* is blank.

- -

Unused field.

Notes

The read operation is not necessary in a standard Mechanical APDL run but is provided as a convenience for those who want to read a coded node file (such as from another mesh generator or from a CAD/CAM program).

Data should be formatted as produced via **NWRITE (p. 1305)**.

Only nodes within the node range specified via **NRRANG (p. 1265)** are read from the file. Duplicate nodes already in the database are overwritten.

The file is rewound before and after reading. Reading continues until the end of the file.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Read Node File

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NREFINE, *NN1*, *NN2*, *NINC*, *LEVEL*, *DEPTH*, *POST*, *RETAIN*

Refines the mesh around specified nodes.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NN1, NN2, NINC

Nodes (*NN1* to *NN2* in increments of *NINC*) around which the mesh is to be refined. *NN2* defaults to *NN1*, and *NINC* defaults to 1. If *NN1* = ALL, *NN2* and *NINC* are ignored and all selected nodes are used for refinement. If *NN1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NN1* (*NN2* and *NINC* are ignored).

LEVEL

Amount of refinement to be done. Specify the value of *LEVEL* as an integer from 1 to 5, where a value of 1 provides minimal refinement, and a value of 5 provides maximum refinement (defaults to 1).

DEPTH

Depth of mesh refinement in terms of number of elements outward from the indicated nodes (defaults to 1).

POST

Type of postprocessing to be done after element splitting, in order to improve element quality:

OFF

No postprocessing will be done.

SMOOTH

Smoothing will be done. Node locations may change.

CLEAN

Smoothing and cleanup will be done. Existing elements may be deleted, and node locations may change (default).

RETAIN

Flag indicating whether quadrilateral elements must be retained in the refinement of an all-quadrilateral mesh. (Mechanical APDL ignores this argument if you are refining anything other than a quadrilateral mesh.)

ON

The final mesh will be composed entirely of quadrilateral elements, regardless of the element quality (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

The final mesh may include some triangular elements to maintain element quality and provide transitioning.

Notes

This command refines all area elements and tetrahedral volume elements adjacent to the specified nodes.

Mesh refinement is not available on a solid model containing initial conditions at nodes (**IC (p. 921)**), coupled nodes (**CP (p. 421)** family of commands), constraint equations (**CE (p. 297)** family of commands), or boundary conditions or loads applied directly to any of its nodes or elements. This restriction applies to nodes and elements anywhere in the model, not just in the region where you want to request mesh refinement. For more information about mesh refinement, see Revising Your Model.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Refine At>Nodes

NRLSUM,*SIGNIF*,*Label*,*LabelCSM*,*ForceType*

Specifies the Naval Research Laboratory (NRL) sum mode combination method.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single-point, multi-point, or DDAM response spectrum (*Spectype* = SPRS, MPRS, or DDAM on **SPOPT (p. 1837)**), the significance level of a mode is defined as the mode coefficient divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT (p. 1837)**,PSD.)

Label

Label identifying the combined mode solution output.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

LabelCSM

Label identifying the CSM (Closely Spaced Modes) method.

CSM

Use the CSM method.

Blank

Do not use the CSM method (default).

ForceType

Label identifying the forces to be combined:

STATIC

Combine the modal static forces (default).

TOTAL

Combine the modal static plus inertial forces.

Notes

This command is also valid in PREP7. This mode combination method is usually used for **SPOPT (p. 1837)**,DDAM.

This CSM method is only applicable in a DDAM analysis (**SPOPT (p. 1837)**,*DDAM*). Element results calculation based on modal element results (*Elcalc* on **SPOPT (p. 1837)**) is not supported and is automatically reset for this method. The CSM method combines two closely spaced modes into one mode when their frequencies are within 10 percent of the common mean frequency and their responses are opposite in sign. The contribution of these closely spaced modes is then included in the NRL sum as a single effective mode. Refer to Closely Spaced Modes (CSM) Method in the *Theory Reference* for more information.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine>NRLSUM Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine>NRLSUM Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine>NRLSUM Method

Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine>NRLSUM Method

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine>NRLSUM Method

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine>NRLSUM Method

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*NRM,*Name*,*NormType*,*ParR*,*Normalize*

Computes the norm of the specified matrix or vector.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name

Matrix or vector for which the norm will be computed. This can be a dense matrix (created by the ***DMAT (p. 551)** command), a sparse matrix (created by the ***SMAT (p. 1801)** command) or a vector (created by the ***VEC (p. 2085)** command)

NormType

Mathematical norm to use:

NRM2 --

L2 (Euclidian or SRSS) norm (default).

NRM1 --

L1 (absolute sum) norm (vectors and dense matrices only).

NRMINF --

Maximum norm.

ParR

Parameter name that contains the result.

Normalize

Normalization key; to be used only for vectors created by ***VEC (p. 2085)**:

YES --

Normalize the vector such that the norm is 1.0.

NO --

Do not normalize the vector (default).

Notes

The NRM2 option corresponds to the Euclidian or L2 norm and is applicable to either vectors or matrices:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\forall x \in \mathbb{R}^n, \|x\|_2 = \left(\sum_{i=1}^{n} |x_i|^2 \right)^{\frac{1}{2}}$

$\forall x \in \mathbb{C}^n, \|x\|_2 = \left(\sum_{i=1}^{n} \overline{x}_I \cdot x_i \right)^{\frac{1}{2}}$ where $\overline{x}_I$ is the complex conjugate of x_i

$\forall A \in \mathbb{R}^{nxn}, \|A\|_2 = \sqrt{\rho\left(A^* A\right)}$ = largest eigenvalue of $\left(A^* A\right)^{\frac{1}{2}}$

The NRM1 option corresponds to the L1 norm and is applicable to vectors and dense matrices:

$\forall x \in \mathbb{R}^n$ or $\mathbb{C}^n, \|x\|_1 = \sum_{i=1}^{n} |x_i|$

$\forall A \in \mathbb{R}^{nxn}$ or $\mathbb{C}^{nxn}, \|A\|_1 = \max_{j \in [i,n]} \sum_{i=1}^{n} |a_{ij}|$

The NRMINF option is the maximum norm and is applicable to either vectors or matrices:

$\forall x \in \mathbb{R}^n$ or $\mathbb{C}^n, \|x\|_\infty = \max_{i \in [i,n]} |x_i|$

$\forall A \in \mathbb{R}^{nxn}$ or $\mathbb{C}^{nxn}, \|A\|_\infty = \max_{i \in [i,n]} \sum_{j=1}^{n} |a_{ij}|$

Menu Paths

This command cannot be accessed from a menu.

NROPT,*Option1*,*Option2*,*Optval*

Specifies the Newton-Raphson options in a static or full transient analysis.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option1

Option key:

AUTO

Let the program choose the option (default).

FULL

Use full Newton-Raphson.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MODI

Use modified Newton-Raphson.

INIT

Use the previously computed matrix (initial-stiffness).

UNSYM

Use full Newton-Raphson with unsymmetric matrices of elements where the unsymmetric option exists.

Option2

Option key:

CRPL

When applicable in a static creep analysis, activates modified Newton-Raphson with a creep-ratio limit. Valid only when *Option1* = AUTO.

Optval

If *Option2* is blank, *Optval* is the Adaptive Descent Key (*Adptky*):

ON

Use adaptive descent (default if frictional contact exists). Explicit ON is valid only if *Option* = FULL.

OFF

Do not use adaptive descent (default in all other cases).

If *Option2* = CRPL, *Optval* is the creep ratio limit:

CRLIMIT

The creep ratio limit for use with the modified Newton-Raphson procedure. Valid only when *Option1* = AUTO (default) and *Option2* = CRPL. Typically, this value should not exceed 0.15 in order to make the modified Newton-Raphson solution converge efficiently. For more information about the creep ratio limit, see the **CUTCONTROL (p. 445)** command.

Notes

The **NROPT (p. 1261)** command specifies the Newton-Raphson option used to solve the nonlinear equations in a static or full transient analysis.

The automatic modified Newton-Raphson procedure with creep-ratio limit control (**NROPT (p. 1261)**,AUTO,CRPL,*CRLIMIT*) applies to static creep analysis only. When the creep ratio is smaller than the value of the creep ratio limit specified, the modified Newton-Raphson procedure is used. If convergence difficulty occurs during solution, use the full Newton-Raphson procedure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The command **NROPT (p. 1261)**,UNSYM is also valid in a linear non-prestressed modal analysis that is used to perform a brake squeal analysis. In this special case, the command is used only to generate the unsymmetric stiffness matrix; no Newton-Raphson iterations are performed.

NROPT (p. 1261),MODI and **NROPT (p. 1261)**,INIT are only applicable with the sparse solver (**EQSLV (p. 648)**,SPARSE). Thermal analyses will always use full Newton-Raphson irrespective of the *Option1* value selected.

See Newton-Raphson Option in the *Structural Analysis Guide* for more information.

This command is also valid in PREP7.

Switching Between the Symmetric and Unsymmetric Option

Normally, switching from the symmetric Newton-Raphson option (**NROPT (p. 1261)**,FULL) to the unsymmetric option (**NROPT (p. 1261)**,UNSYM) or from the unsymmetric option to the symmetric option is allowed between load steps within the same analysis type. This is applicable to linear and nonlinear, static and full transient analyses.

Under the following circumstances, the solution could be slightly different or inaccurate if you switch from symmetric to unsymmetric or vice versa:

- The underlying elements or materials are unsymmetric by their mathematical definition, and you switch from unsymmetric to symmetric.
- You change analysis types and also switch from symmetric to unsymmetric (or vise versa) at the same time. This situation could result in failures such as data corruption or a core dump and should therefore be avoided.
- In some rare cases, switching between the symmetric and unsymmetric options can cause a system core dump when reading/writing the `.esav` or `.osav` file, and the analysis terminates. Typically, this happens when the record length of the element nonlinear saved variables cannot be altered between load steps by their mathematical definition.

If all the elements and the material are symmetric by their mathematical definition and you use the unsymmetric option, the solution accuracy is the same as the symmetric option. However, the analysis will run twice as slow as the symmetric case.

In a static or full transient linear perturbation base analysis, be aware that if the unsymmetric Newton-Raphson procedure is used, you must specify the UNSYM or DAMP eigensolver option in the downstream modal analysis, which may be more expensive than symmetric modal analysis.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NROTAT, *NODE1*, *NODE2*, *NINC*

Rotates nodal coordinate systems into the active system.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1*, *NODE2*, *NINC

Rotate nodes from *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and all selected nodes (**NSEL (p. 1266)**) are rotated. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Command Default

No automatic nodal rotation.

Notes

Rotates nodal coordinate systems into the active coordinate system. Nodal coordinate systems may be automatically rotated into the active (global or local) coordinate system as follows: Rotations in Cartesian systems will have nodal x directions rotated parallel to the Cartesian X direction. Rotations in cylindrical, spherical or toroidal systems will have the nodal x directions rotated parallel to the R direction. Nodes at (or near) a zero radius location should not be rotated. Nodal coordinate directions may be displayed (**/PSYMB (p. 1552)**). Nodal forces and constraints will also appear rotated when displayed if the nodal coordinate system is rotated.

Note:

When the nodal coordinate systems are defined, they remain parallel to the global Cartesian system unless subsequently rotated.

Previously specified rotations on the specified nodes are overridden.

See the **NMODIF (p. 1247)**, **NANG (p. 1208)**, and **NORA (p. 1252)** commands for other rotation options.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>RotateNode>To Active CS

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Active CS

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NRRANG, *NMIN*, *NMAX*, *NINC*

Specifies the range of nodes to be read from the node file.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NMIN*, *NMAX*, *NINC

Node range is defined from *NMIN* (defaults to 1) to *NMAX* (defaults to 999999999) in steps of *NINC* (defaults to 1).

Notes

Defines the range of nodes to be read (**NREAD (p. 1256)**) from the node file. Also implies an element range since only elements fully attached to these nodes will be read from the element file.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Read Node File

NSCALE, *INC*, *NODE1*, *NODE2*, *NINC*, *RX*, *RY*, *RZ*

Generates a scaled set of nodes from a pattern of nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

INC

Do this scaling operation one time, incrementing all nodes in the given pattern by *INC*. If *INC* = 0, nodes will be redefined at the scaled locations.

NODE1*, *NODE2*, *NINC

Scale nodes from pattern of nodes beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and pattern is all selected nodes (**NSEL (p. 1266)**). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

RX*, *RY*, *RZ

Scale factor ratios. Scaling is relative to the origin of the active coordinate system (RR, Rθ, RZ for cylindrical, RR, Rθ, RΦ for spherical or toroidal). If absolute value of ratio > 1.0, pattern is enlarged. If < 1.0, pattern is reduced. Ratios default to 1.0 (each).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates a scaled pattern of nodes from a given node pattern. Scaling is done in the active coordinate system. Nodes in the pattern may have been generated in any coordinate system.

This command is also valid in the **/MAP (p. 1107)** processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Nodes>Scale & Copy

Main Menu>Preprocessor>Modeling>Move / Modify>Nodes>Scale & Move

Main Menu>Preprocessor>Modeling>Operate>Scale>Nodes>Scale & Copy

Main Menu>Preprocessor>Modeling>Operate>Scale>Nodes>Scale & Move

---

NSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KABS*

Selects a subset of nodes.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STAT

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item

Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to NODE.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

VMIN

Minimum value of item range. Ranges are node numbers, set numbers, coordinate values, load values, or result values as appropriate for the item. A component name (as specified on the **CM (p. 356)** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored).

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN* for input values. For result values, *VMAX* defaults to infinity if *VMIN* is positive, or to zero if *VMIN* is negative.

VINC

Value increment within range. Used only with integer ranges (such as for node and set numbers). Defaults to 1. *VINC* cannot be negative.

KABS

Absolute value key:

0

Check sign of value during selection.

1

Use absolute value during selection (sign ignored).

Command Default

All nodes are selected.

Notes

Selects a subset of nodes. For example, to select a new set of nodes based on node numbers 1 through 7, use **NSEL (p. 1266)**,S,NODE,,1,7. The subset is used when the ALL label is entered (or implied) on other

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

commands, such as **NLIST (p. 1239)**,ALL. Only data identified by node number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

When selecting nodes by results, the full graphics value is used, regardless of whether PowerGraphics is on.

Solution result data consists of two types, 1) nodal degree of freedom--results initially calculated at the nodes (such as displacement, temperature, pressure, etc.), and 2) element--results initially calculated elsewhere (such as at an element integration point or thickness location) and then recalculated at the nodes (such as stresses, strains, etc.). Various element results also depend upon the recalculation method and the selected results location (**AVPRIN (p. 210)**, **RSYS (p. 1639)**, **FORCE (p. 772)**, **LAYER (p. 1015)** and **SHELL (p. 1784)**).

You must have all the nodes (corner and midside nodes) on the external face of the element selected to use *Item* = EXT.

This command is valid in any processor.

For Selects based on non-integer numbers (coordinates, results, etc.), items that are within the range VMIN-*Toler* and VMAX+*Toler* are selected. The default tolerance *Toler* is based on the relative values of VMIN and VMAX as follows:

- If VMIN = VMAX, *Toler* = 0.005 x VMIN.
- If VMIN = VMAX = 0.0, *Toler* = 1.0E-6.
- If VMAX ≠ VMIN, *Toler* = 1.0E-8 x (VMAX-VMIN).

Use the **SELTOL (p. 1710)** (p. 1710) command to override this default and specify *Toler* explicitly.

Table 198: NSEL (p. 1266) - Valid Item and Component Labels

Valid Item and Component Labels NSEL (p. 1266), *Type,Item,Comp,VMIN,VMAX,VINC,KABS*

Valid item and component labels for input values are:

Item	Comp	Description
NODE		Node number.
EXT		Nodes on exterior of selected elements (ignore remaining fields).
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle.
M		Master node number.
CP		Coupled set number.
CE		Constraint equation set number.
D	U	Any of X, Y, or Z structural displacements. Amplitude only, if complex.
"	UX, UY, UZ	X, Y, or Z structural displacement. Amplitude only, if complex.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Item and Component Labels NSEL (p. 1266),
Type,Item,Comp,VMIN,VMAX,VINC,KABS

Valid item and component labels for input values are:

Item	Comp	Description
"	ROT	Any of X, Y, or Z structural rotations. Amplitude only, if complex.
"	ROTX, ROTY, ROTZ	X, Y, or Z structural rotation. Amplitude only, if complex.
"	TEMP, TBOT, TE2, TE3, . . ., TTOP	Temperature.
"	PRES	Pressure (for example, PRES degree of freedom for acoustic elements).
"	VOLT	Electric potential.
"	MAG	Magnetic scalar potential.
"	V	Any of X, Y, or Z fluid velocities.
"	VX, VY, VZ	X, Y, or Z fluid velocity.
"	AZ	2D magnetic vector potential. Amplitude only, if complex.
"	CURR	Current.
"	EMF	Electromotive force drop.
F	F	Any of X, Y, or Z structural forces. Amplitude only, if complex.
"	FX, FY, FZ	X, Y, or Z structural force. Amplitude only, if complex.
"	M	Any of X, Y, or Z structural moments. Amplitude only, if complex
"	MX, MY, MZ	X, Y, or Z structural moment. Amplitude only, if complex.
"	HEAT, HBOT, HE2, HE3, . . ., HTOP	Heat flow.
"	FLOW	Fluid flow.
"	AMPS	Current flow.
"	FLUX	Magnetic flux.
"	CSGZ	Magnetic current segment component. Amplitude only, if complex.
	RATE	Diffusion flow rate.
"	CHRG	Electric charge.
"	CHRGD	Electric charge density.
BF	TEMP	Nodal temperature.
"	FLUE	Nodal fluence.
"	HGEN	Nodal heat generation rate.
"	JS	Any of X, Y, or Z current densities. Amplitude only, if complex.
"	JSX, JSY, JSZ	X, Y, or Z current density. Amplitude only, if complex.
"	MVDI	Magnetic virtual displacements flag.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Item and Component Labels NSEL (p. 1266),
Type,Item,Comp,VMIN,VMAX,VINC,KABS

Valid item and component labels for input values are:

Item	Comp	Description
"	DGEN	Nodal diffusing substance generation rate.

Table 199: NSEL - Valid Item and Component Labels for Nodal DOF Result Values

Item	Comp	Description
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CONC		Concentration.
CURR		Current.
EMF		Electromotive force drop.

Table 200: NSEL - Valid Item and Component Labels for Element Result Values

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1,2,3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1,2,3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1,2,3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
EPSW		Swelling strain.
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain.
"	1, 2, 3	Principal diffusion strain.
"	INT, EQV	Diffusion strain intensity or diffusion equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
CONT	STAT[1]	Contact status.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum
FMAG	X, Y, Z, SUM	Component electromagnetic forces or vector sum.

1. For more information on the meaning of contact status and its possible values, see Reviewing Results in POST1 in the *Contact Technology Guide*.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Delete Elements

Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts

Utility Menu>Select>Entities

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NSLA, *Type*, *NKEY*

Selects those nodes associated with the selected areas.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of node select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

NKEY

Specifies whether only interior area nodes are to be selected:

0

Select only nodes interior to selected areas.

1

Select all nodes (interior to area, interior to lines, and at keypoints) associated with the selected areas.

Notes

Valid only if the nodes were generated by an area meshing operation (**AMESH (p. 121)**, **VMESH (p. 2140)**) on a solid model that contains the selected areas.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NSLE, *Type*, *NodeType*, *Num*

Selects those nodes attached to the selected elements.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of node select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

NodeType

Label identifying type of nodes to consider when selecting:

ALL

Select all nodes of the selected elements (default).

ACTIVE

Select only the active nodes. An active node is a node that contributes DOFs to the model.

INACTIVE

Select only inactive nodes (such as orientation or radiation).

CORNER

Select only corner nodes.

MID

Select only midside nodes.

POS

Select nodes in position *Num*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FACE

Select nodes on face *Num*.

Num

Position or face number for *NodeType* = POS or FACE.

Notes

NSLE (p. 1273) selects *NodeType* nodes attached to the currently-selected set of elements. Only nodes on elements in the currently-selected element set can be selected.

Note:

When using degenerate hexahedral elements, **NSLE (p. 1273)**, *U*,*CORNER* and **NSLE (p. 1273)**,*S*,*MID* will not select the same set of nodes because some nodes appear as both corner and midside nodes.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>CMS>CMS Superelements>By Picking

Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts

Utility Menu>Select>Entities

NSLK, *Type*

Selects those nodes associated with the selected keypoints.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of node select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

U

Unselect a set from the current set.

Notes

Valid only if the nodes were generated by a keypoint meshing operation (**KMESH (p. 985)**, **LMESH (p. 1057)**, **AMESH (p. 121)**, **VMESH (p. 2140)**) on a solid model that contains the selected keypoints.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

NSLL, *Type*, *NKEY*

Selects those nodes associated with the selected lines.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of node select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

NKEY

Specifies whether only interior line nodes are to be selected:

0

Select only nodes interior to selected lines.

1

Select all nodes (interior to line and at keypoints) associated with the selected lines.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Valid only if the nodes were generated by a line meshing operation (**LMESH (p. 1057)**, **AMESH (p. 121)**, **VMESH (p. 2140)**) on a solid model that contains the associated lines.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

NSLV, *Type*, *NKEY*

Selects those nodes associated with the selected volumes.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of node select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

NKEY

Specifies whether only interior volume nodes are to be selected:

0

Select only nodes interior to selected volumes.

1

Select all nodes (interior to volume, interior to areas, interior to lines, and at keypoints) associated with the selected volumes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Valid only if the nodes were generated by a volume meshing operation (**VMESH (p. 2140)**) on a solid model that contains the selected volumes.

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

NSMASS,*Elem*,*Action*,*UNIT*,*VAL*,*DISTRB*

Applies nonstructural mass to selected elements.

PREP7 (p. 22): Materials (p. 24)
SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Elem

Element on which to apply nonstructural mass. If ALL, the command applies nonstructural mass to *all* selected elements (**ESEL (p. 661)**). A component name is also valid.

Action

Command action requested:

ADD – Add nonstructural mass to the specified element or elements.

DEL – Delete all previously defined nonstructural mass from selected element(s).

LIST – List the input value of nonstructural mass. If multiple **NSMASS (p. 1277)** commands are issued, this option lists the total accumulated nonstructural mass.

The following options apply only when *Action* = ADD:

UNIT

Unit of the input value of nonstructural mass:

0 – Total Mass (default).

1 – Mass per unit volume.

2 – Mass per unit area.

3 – Mass per unit length.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL

Input value of the nonstructural mass. A positive value increases (accumulates) the total nonstructural mass. A negative value reduces the total nonstructural mass.

DISTRB

Nonstructural mass distribution method. Valid only when *UNIT* = 0.

0 – Mass proportional distribution (default).

1 – Volume proportional distribution.

2 – Area proportional distribution.

3 – Length proportional distribution.

Notes

NSMASS (p. 1277) supports current-technology structural elements only:

Table 201: Supported Elements

Category	Element Types
2D/3D Structural Solid	PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SOLID285
2D/3D Structural Shell	SHELL208, SHELL209, SHELL181, SHELL281
2D/3D Surface Effect	SURF153, SURF154, SURF156
2D/3D Structural Link, Beam, Pipe, and Elbow	LINK180, CABLE280, BEAM188, BEAM189, PIPE288, PIPE289, ELBOW290
2D/3D Gasket	INTER192, INTER193, INTER194, INTER195

Applying Nonstructural Mass

NSMASS (p. 1277) offers two methods for nonstructural mass application:

- Distribution of total nonstructural mass (p. 1279)
- Direct input of nonstructural mass density (p. 1279)

For consecutive accumulation or reduction of total nonstructural mass to the same elements, you can issue multiple **NSMASS (p. 1277)** commands, each specifying a different unit (*UNIT*) or distribution method (*DISTRB*).

Use care when selecting the unit and distribution method to ensure compatibility with the selected elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distribution of Total Nonstructural Mass (*UNIT* = 0)

Specify *UNIT* = 0 to distribute the total nonstructural mass over all selected elements proportionally to the mass, volume, area, or length (*DISTRB* = 0, 1, 2, or 3, respectively):

Table 202: Supported Total Mass Distribution Methods

DISTRB	Applicable Element Types
0, 1	Supports all element types listed in Table 201: Supported Elements (p. 1278) with the exception of SURF156.
2	Supports these structural shell and surface-effect elements: SHELL208, SHELL209, SHELL181, SHELL281, SURF153, and SURF154.
3	Supports structural link, beam, pipe, and elbow elements, and the SURF156 structural surface line-load-effect element.

Direct Input of Nonstructural Mass Density (*UNIT* = 1, 2, or 3)

The nonstructural mass is applied directly to the selected element in one form of mass densities: mass per unit volume (*UNIT* = 1), mass per unit area (*UNIT* = 2), or mass per unit length (*UNIT* = 3):

Table 203: Supported Units for Direct Input of Mass Density

DISTRB	Applicable Element Types
1	Supports all element types listed in Table 201: Supported Elements (p. 1278) with the exception of SURF156.
2	Supports these structural shell and surface-effect elements: SHELL208, SHELL209, SHELL181, SHELL281, SURF153, and SURF154.
3	Supports structural link, beam, pipe, and elbow elements, and the SURF156 structural surface line-load-effect element.

Nonstructural Mass Behavior: Saturated Mass vs. Attached Mass

NSMASS (p. 1277) accounts for two behaviors of the nonstructural mass:

- **The nonstructural mass is applied uniformly to the entire volume of the element (that is, the saturated nonstructural mass).**

  Example: The moisture content in a structure can be simulated with the saturated nonstructural mass.

  This behavior of the nonstructural mass can be defined via a mass- or volume-proportional distribution of total nonstructural mass (*DISTRB* = 0 or 1), or direct input of mass density (*UNIT* = 1).

- **The nonstructural mass is attached to the element at the location where element nodes are defined (that is, at the attached nonstructural mass).**

  Example: The thin exterior insulation on a pipe can be simulated with the attached nonstructural mass.

  This behavior of the nonstructural mass can be defined via area or length proportional distribution of total nonstructural mass (*DISTRB* = 2 or 3), or via direct input of mass density (*UNIT* = 2 or 3).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The attached nonstructural mass is assumed to have a negligible thickness or cross-sectional area. Therefore, for structural elements with rotational degrees of freedom, element rotational inertia is not affected by the attached nonstructural mass. To properly account for the attached nonstructural mass on these elements, position the element nodes (**SECOFFSET (p. 1685)**) at the locations where the nonstructural mass is applied (for example, at the top surface of a shell element).

Assumptions and Limitations

- Elements with inhomogeneous materials (such as a structural shell with a composite section) are not supported for mass proportional distribution (*DISTRIB* = 0).
- Elements with a TAPER, GENB, COMB, or GENS section type are not supported.
- The element volume, area, or length used for total nonstructural mass distribution (*DISTRIB* = 0, 1, 2, or 3 respectively) are calculated using the undeformed element geometry.
- The element mass used for total nonstructural mass distribution (*DISTRB* = 0) is calculated using undeformed element geometry and mass density at the reference temperature.
- **NSMASS (p. 1277)** does not account for subsequent modifications made to the selected elements. Issue **NSMASS (p. 1277)** only *after* defining the selected elements.

Menu Paths

This command cannot be accessed from a menu.

NSMOOTH, *NPASS*

Smooths selected nodes among selected elements.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NPASS

Number of smoothing passes. Defaults to 3.

Notes

Repositions each selected node at the average position of its immediate neighbors on the selected elements. The node positions converge after some number of smoothing passes. For some initial conditions, *NPASS* may need to be much larger than 3. If the boundary of a mesh is to be undisturbed (usually desirable), the boundary nodes should be unselected before issuing **NSMOOTH (p. 1280)**.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NSOL, *NVAR*, *NODE*, *Item*, *Comp*, *Name*, *SECTOR*

Specifies nodal data to be stored from the results file.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number assigned to this variable. Variable numbers can be 2 to *NV* (**NUMVAR (p. 1302)**). Overwrites any existing results for this variable.

NODE

Node for which data are to be stored.

Item

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

Name

Thirty-two character name identifying the item on printouts and displays. Defaults to a label formed by concatenating the first four characters of the *Item* and *Comp* labels.

SECTOR

For a full harmonic cyclic symmetry solution, the sector number for which the results from NODE are to be stored.

Notes

Stores nodal degree of freedom and solution results in a variable. For more information, see Data Interpreted in the Nodal Coordinate System in the *Modeling and Meshing Guide*.

For *SECTOR*>1, the result is in the nodal coordinate system of the base sector, and it is rotated to the expanded sector's location. Refer to Using the /CYCEXPAND Command in the *Cyclic Symmetry Analysis Guide* for more information.

Table 204: NSOL (p. 1281) - Valid Item and Component Labels

Valid Item and Component Labels NSOL (p. 1281), *NVAR*, *NODE*, *Item*, *Comp*, *Name*

Valid item and component labels for nodal degree of freedom results are:

Item	Comp	Description
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Item and Component Labels NSOL (p. 1281), *NVAR, NODE, Item, Comp, Name*

Valid item and component labels for nodal degree of freedom results are:

Item	Comp	Description
TEMP[1]		Temperature.
PRES		Pressure.
GFV1, GFV2, GFV3		Nonlocal field values 1, 2, and 3.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z	X, Y, or Z fluid velocity in a fluid analysis.
A	X, Y, Z	X, Y, or Z magnetic vector potential in an electromagnetic analysis.
CONC		Concentration.
VEL	X, Y, Z	X, Y, or Z velocity in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
ACC	X, Y, Z	X, Y, or Z acceleration in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
OMG	X, Y, Z	X, Y, or Z rotational velocity in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
DMG	X, Y, Z	X, Y, or Z rotational acceleration in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
CURR		Current.
EMF		Electromotive force drop.
SPL		Sound pressure level.
SPLA		A-weighted sound pressure level (dBA).
ENKE		Acoustic energy density

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NSORT, *Item*, *Comp*, *ORDER*, *KABS*, *NUMB*, *SEL*

Sorts nodal data.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item to be sorted on. Valid item labels are shown in the table below. Some items also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

ORDER

Order of sort operation:

0

Sort into descending order.

1

Sort into ascending order.

KABS

Absolute value key:

0

Sort according to real value.

1

Sort according to absolute value.

NUMB

Number of nodal data records to be sorted in ascending or descending order (*ORDER*) before sort is stopped (remainder will be in unsorted sequence) (defaults to all nodes).

SEL

Allows selection of nodes in the sorted field.

(blank)

No selection (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SELECT

Select the nodes in the sorted list.

Command Default

Use ascending node-number order.

Notes

Values are in the active coordinate system (**CSYS (p. 441)** for input data or **RSYS (p. 1639)** for results data). Various element results also depend upon the recalculation method and the selected results location (**AVPRIN (p. 210)**, **RSYS (p. 1639)**, **SHELL (p. 1784)**, **ESEL (p. 661)**, and **NSEL (p. 1266)**). If simultaneous load cases are stored, the last sorted sequence formed from any load case applies to all load cases. Use **NUSORT (p. 1303)** to restore the original order. This command is not valid with PowerGraphics.

Table 205: NSORT (p. 1283) - Valid Item and Component Labels

Valid Item and Component Labels NSORT (p. 1283), *Item,Comp,ORDER,KABS,NUMB,SEL*

Valid item and component labels for input values are:

Item	Comp	Description
LOC	X, Y, Z	X, Y, or Z location.
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle.

Table 206: NSORT (p. 1283) - Valid Item and Component Labels for Nodal DOF Result Values

Item	Comp	Description
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP		Temperature (includes TEMP, TBOT, TE2, TE3, . . ., TTOP values).
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CONC		Concentration
CURR		Current.
EMF		Electromotive force drop.

Table 207: NSORT (p. 1283) - Valid Item and Component Labels for Element Result Values

Item	Comp	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2,3	Principal stress.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain.
"	1, 2, 3	Principal diffusion strain.
"	INT, EQV	Diffusion strain intensity or diffusion equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
FAIL	MAX	Maximum of all active failure criteria defined at the current location. (See the **FCTYP (p. 740)** command for details.) [1]
"	EMAX	Maximum Strain Failure Criterion [1]
"	SMAX	Maximum Stress Failure Criterion [1]
"	TWSI	Tsai-Wu Strength Index Failure Criterion [1]
"	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion [1]
"	HFIB	Hashin Fiber Failure Criterion. [1][2]
"	HMAT	Hashin Matrix Failure Criterion. [1][2]
"	PFIB	Puck Fiber Failure Criterion. [1][2]
"	PMAT	Puck Matrix Failure Criterion. [1][2]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
"	USR1, USR2, ..., USR9	User-defined failure criteria [1][2][3]
CONT	STAT [4]	Contact status.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum
FMAG	X, Y, Z, SUM	Component electromagnetic forces or vector sum.

1. Works only if failure criteria information is provided. (For more information, see the documentation for the **FC (p. 732)** and **TB (p. 1899)** commands.)
2. Must be added via the **FCTYP (p. 740)** command first.
3. Works only if user failure criteria routine is provided.
4. For more information about contact status and its possible values, see Reviewing Results in POST1 in the *Contact Technology Guide*.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes

Utility Menu>Parameters>Get Scalar Data

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NSTORE, *TINC*

Defines which time points are to be stored.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TINC

Store data associated with every *TINC* time (or frequency) point(s), within the previously defined range of *TMIN* to *TMAX* (**TIMERANGE (p. 2001)**). (Defaults to 1)

Command Default

Store every point.

Notes

Defines which time (or frequency) points within the range are to be stored.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

NSUBST, *NSBSTP*, *NSBMX*, *NSBMN*, *Carry*

Specifies the number of substeps to be taken this load step.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSBSTP

Number of substeps to be used for this load step (that is, the time step size or frequency increment). If automatic time stepping is used (**AUTOTS (p. 207)**), *NSBSTP* defines the size of the first substep.

If contact elements (TARGE169, TARGE170, CONTA172, CONTA174, CONTA175, or CONTA177) are used, defaults to 1 or 20 substeps, depending on the physics of the model. If none of these contact elements are used, defaults to 1 substep.

NSBMX

Maximum number of substeps to be taken (that is, the minimum time step size) if automatic time stepping is used. The program automatically determines the default based on the physics of the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NSBMN

Minimum number of substeps to be taken (that is, the maximum time step size) if automatic time stepping is used. The program automatically determines the default based on the physics of the model.

Carry

Time step carryover key (program-determined default depending on the problem physics):

OFF

Use *NSBSTP* to define time step at start of each load step.

ON

Use final time step from previous load step as the starting time step (if automatic time stepping is used).

The program automatically determines the default based on the physics of the model.

Notes

See **DELTIM (p. 515)** for an alternative input. It is recommended that all fields of this command be specified for solution efficiency and robustness.

When the arc-length method is active (**ARCLEN (p. 175)** command), the *NSBMX* and *NSBMN* arguments are ignored.

Changing the number of substeps upon restarting an analysis during a load step is not recommended. You should only change the number of substeps between load steps.

For a harmonic balance method (HBM) analysis, use *NSBSTP* to specify the number of substeps when no continuation method is used (*CorrMeth* = OFF on **HBMOPT (p. 890)**,CONTSET). The first substep is at *FREQB* + (*FREQE* - *FREQB*)/*NSBSTP* as in other harmonic analyses, where *FREQB* and *FREQE* are the beginning and ending frequencies of a solution specified with the **HARFRQ (p. 885)** command. *NSBSTP* is the only argument used for HBM analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Freq and Substeps

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>HOWP and Substeps

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substeps

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Load Step Opts>Time/Frequenc>Freq and Substeps

Main Menu>Solution>Load Step Opts>Time/Frequenc>HOWP and Substeps

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substeps

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NSVR, *ITYPE*, *NSTV*

Defines the number of variables for user-programmable element options.

PREP7 (p. 22): Element Type (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITYPE

Element type number as defined on the **ET (p. 686)** command.

NSTV

Number of extra state variables to save (must be no more than 840).

Command Default

No extra variables are saved.

Notes

Defines the number of extra variables that need to be saved for user-programmable (system-dependent) element options, for example, material laws through user subroutine USERPL. *ITYPE* must first be defined with the **ET (p. 686)** command.

Menu Paths

This command cannot be accessed from a menu.

NSYM, *Ncomp*, *INC*, *NODE1*, *NODE2*, *NINC*

Generates a reflected set of nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ncomp

Symmetry key:

X

X (or R) symmetry (default).

Y

Y (or θ) symmetry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z

Z (or Φ) symmetry.

INC

Increment all nodes in the given pattern by *INC* to form the reflected node pattern.

NODE1*, *NODE2*, *NINC

Reflect nodes from pattern beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and pattern is all selected nodes (**NSEL (p. 1266)**). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Generates nodes from a given node pattern by a symmetry reflection. Reflection is done in the active coordinate system by changing a particular coordinate sign. Nodes in the pattern may have been generated in any coordinate system. Nodal rotation angles are not reflected.

Symmetry reflection may be used with any node pattern, in any coordinate system, as many times as desired. Reflection is accomplished by a coordinate sign change (in the active coordinate system). For example, an X-reflection in a Cartesian coordinate system generates additional nodes from a given pattern, with a node increment added to each node number, and an X coordinate sign change. An R-reflection in a cylindrical coordinate system gives a reflected "radial" location by changing the "equivalent" Cartesian (that is, the Cartesian system with the same origin as the active cylindrical system) X and Y coordinate signs. An R-reflection in a spherical coordinate system gives a reflected "radial" location by changing the equivalent Cartesian X, Y, and Z coordinate location signs. Nodal coordinate system rotation angles are not reflected.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Nodes

/NUMBER, *NKEY*

Specifies whether numbers, colors, or both are used for displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NKEY

Numbering style:

0

Color (terminal dependent) the numbered items and show numbers.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Color the numbered items. Do not show the numbers.

2

Show the numbers. Do not color the items.

-1

Do not color the items or show the numbers. For contour plots, the resulting display will vary (see below).

Notes

Specifies whether numbers, colors, or both are used for numbering displays (**/PNUM (p. 1457)**) of nodes, elements, keypoints, lines, areas, and volumes.

Shading is also available for terminals configured with more than 4 color planes (**/SHOW (p. 1785)**). Color automatically appears for certain items and may be manually controlled (off or on) for other items.

When you suppress color (*NKEY* = -1) your contour plots will produce different results, depending on your graphics equipment. For non-3D devices (X11, Win32, etc.) your contour plot will be white (no color). For 3D devices, such as OpenGL, the resulting display will be in color.

The following items are automatically given discrete colors: Boundary condition symbols (**/PBC (p. 1356)**), curves on graph displays, and distorted geometry on postprocessing displays. Contour lines in postprocessing displays are automatically colored based upon a continuous, rather than a discrete, spectrum so that red is associated with the highest contour value. On terminals with raster capability (**/SHOW (p. 1785)**), the area between contour lines is filled with the color of the higher contour.

Explicit entity colors or the discrete color mapping may be changed with the **/COLOR (p. 400)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Numbering

NUMCMP, *Label*

Compresses the numbering of defined items.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Items to be compressed:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NODE

Node numbers

ELEM

Element numbers

KP

Keypoint numbers

LINE

Line numbers

AREA

Area numbers

VOLU

Volume numbers

MAT

Material numbers

TYPE

Element type numbers

REAL

Real constant numbers

CP

Coupled set numbers

SECN

Section numbers

CE

Constraint equation numbers

CSYS

Coordinate system numbers

ALL

All item numbers

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **NUMCMP (p. 1291)** command effectively compresses out unused item numbers by renumbering all the items, beginning with one and continuing throughout the model. The renumbering order follows the initial item numbering order (that is, compression lowers the maximum number by "sliding" numbers down to take advantage of unused or skipped numbers). All defined items are renumbered, regardless of whether or not they are actually used or selected. Applicable related items are also checked for re-numbering as described for the merge operation (**NUMMRG (p. 1295)**).

Compressing material numbers (**NUMCMP (p. 1291)**,ALL or **NUMCMP (p. 1291)**,MAT) does *not* update the material number referenced by either of the following:

- A temperature-dependent convection or surface-to-surface radiation load (**SF (p. 1733)**, **SFE (p. 1761)**, **SFL (p. 1774)**, **SFA (p. 1739)**)
- Real constants for multi-material elements

Compression is usually not required unless memory space is limited and there are large gaps in the numbering sequence.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Compress Numbers

NUMEXP, *NUM*, *BEGRNG*, *ENDRNG*, *Elcalc*

Specifies solutions to be expanded from mode-superposition analyses or substructure analyses.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUM

The number of solutions to expand. This value is required.

Num

Number of solutions to expand.

ALL

Expand all substeps between *BEGRNG* and *ENDRNG* (provided that *ENDRNG* > 0). If *BEGRNG* and *ENDRNG* have no specified values, this option expands *all* substeps of *all* load steps.

BEGRNG, ENDRNG

Beginning and ending time (or frequency) range for expanded solutions. The default is 0 for both values.

Elcalc

The element-calculation key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

YES

Calculate element results, nodal loads, and reaction loads (default).

NO

Do not calculate these items.

Command Default

If this command is not issued, the default behavior is to calculate element results, nodal loads, and reaction loads (*Elcalc* = YES).

Notes

Issuing this command with no arguments is invalid. You must specify the number of solutions, or all solutions, to expand (*NUM*).

Specifies a range of solutions to be expanded from analyses that use mode-superposition methods (**ANTYPE (p. 162)**,HARMIC or TRANS) or substructuring (**ANTYPE (p. 162)**,SUBSTR).

For **ANTYPE (p. 162)**,TRANS, *NUM*, evenly spaced solutions are expanded between time *BEGRNG* and time *ENDRNG*.

For **ANTYPE (p. 162)**,HARMIC, *NUM*, evenly spaced solutions are expanded between frequency *BEGRNG* and frequency *ENDRNG*.

The first expansion in all cases is done at the first point beyond *BEGRNG* (that is, at *BEGRNG* + (*ENDRNG* - *BEGRNG*) / *NUM*)).

The resulting results file will maintain the same load step, substep, and time (or frequency) values as the use pass.

For a single expansion of a solution, or for multiple expansions when the solutions are not evenly spaced (such as in a mode-superposition harmonic analysis with the cluster option), Ansys, Inc. recommends issuing one or more **EXPSOL (p. 721)** commands.

NUMEXP (p. 1293) is invalid in these cases:

- In a substructing analysis (**ANTYPE (p. 162)**,*SUBST*) when a factorized matrix file (the .LN22 file generated by the sparse solver) does not exist, causing Mechanical APDL to use the full-resolve method.
- If the full-resolve option is selected (**SEOPT (p. 1715)**).

In both situations, issue **EXPSOL (p. 721)** to perform a single expansion for each solution desired.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Range of Solu's

---

NUMMRG, *Label*, *TOLER*, *GTOLER*, *Action*, *Switch*

Merges coincident or equivalently defined items.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Items to be merged:

NODE

Nodes

ELEM

Elements

KP

Keypoints (will also merge lines, areas, and volumes)

MAT

Materials

TYPE

Element types

REAL

Real constants

SECT

Section types

CP

Coupled sets

CE

Constraint equations

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

All items

TOLER

Range of coincidence. For *Label* = NODE and KP, defaults to 1.0E-4 (based on maximum Cartesian coordinate difference between nodes or keypoints). For *Label* = MAT, REAL, SECT, and CE, defaults to 1.0E-7 (based on difference of the values normalized by the values). Only items within range are merged. (For keypoints attached to lines, further restrictions apply. See the *GTOLER* field and *Merging Solid Model Entities* below.)

GTOLER

Global solid model tolerance -- used only when merging keypoints attached to lines. If specified, *GTOLER* will override the internal relative solid model tolerance. See *Merging Solid Model Entities* below.

Action

Specifies whether to merge or select coincident items.

SELE

Select coincident items but do not merge. *Action* = SELE is only valid for *Label* = NODE.

(Blank)

Merge the coincident items (default).

Switch

Specifies whether the lowest or highest numbered coincident item is retained after the merging operation. This option does not apply to keypoints; that is, for *Label* = KP, the lowest numbered keypoint is retained regardless of the *Switch* setting.

LOW

Retain the lowest numbered coincident item after the merging operation (default).

HIGH

Retain the highest numbered coincident item after the merging operation.

Notes

The **NUMMRG (p. 1295)** command does not change a model's geometry, only the topology.

After issuing the command, the area and volume sizes (**ASUM (p. 203)** and **VSUM (p. 2167)**) may give slightly different results. In order to obtain the same results as before, use **/FACET (p. 731)**, **/NORMAL (p. 1253)**, and **ASUM (p. 203)** / **VSUM (p. 2167)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The merge operation is useful for tying separate but coincident parts of a model together. If not all items are to be checked for merging, use the select commands (**NSEL (p. 1266)**, **ESEL (p. 661)**, etc.) to select items. Only selected items are included in the merge operation for nodes, keypoints, and elements.

By default, the merge operation retains the lowest numbered coincident item. Higher numbered coincident items are deleted. Set *Switch* to HIGH to retain the highest numbered coincident item after the merging operation. Applicable related items are also checked for deleted item numbers and if found, are replaced with the retained item number. For example, if nodes are merged, element connectivities (except superelements), mesh item range associativity, coupled degrees of freedom, constraint equations, master degrees of freedom, degree of freedom constraints, nodal-force loads, nodal surface loads, and nodal body force loads are checked. Merging material numbers (**NUMMRG (p. 1295)**,ALL or **NUMMRG (p. 1295)**,MAT) does *not* update the material number referenced:

- By temperature-dependent film coefficients as part of convection load or a temperature-dependent emissivity as part of a surface-to-surface radiation load (**SF (p. 1733)**, **SFE (p. 1761)**, **SFL (p. 1774)**, **SFA (p. 1739)**)
- By real constants for multi-material elements

When merging tapered beam or pipe sections, the program first uses the associated end sections for merging. If the merge is successful, the program replaces the tapered section database with the end section data.

If a unique load is defined among merged nodes, the value is kept and applied to the retained node. If loads are not unique (not recommended), only the value on the lowest node (or highest if *Switch* = HIGH) is kept (except for "force" loads for which the values are summed if they are not defined via tabular boundary conditions).

Note:

The unused nodes (not recommended) in elements, couplings, constraint equations, etc. may become active after the merge operation.

The *Action* field provides the option of visualizing the coincident items before the merging operation.

Caution:

When merging entities in a model that has already been meshed, the order in which you issue multiple **NUMMRG (p. 1295)** commands is significant. To merge two adjacent meshed regions having coincident nodes and keypoints, always merge nodes (**NUMMRG (p. 1295)**,NODE) *before* merging keypoints (**NUMMRG (p. 1295)**,KP); otherwise, some of the nodes may lose their association with the solid model (causing other operations to fail). To prevent mesh failure, avoid multiple merging and meshing operations .

After a **NUMMRG (p. 1295)**,NODE command executes, some nodes may be attached to more than one solid entity. As a result, subsequent attempts to transfer solid model loads to the elements may not be successful. Issue **NUMMRG (p. 1295)**,KP to correct this problem. Do *NOT* issue **VCLEAR (p. 2076)** before issuing **NUMMRG (p. 1295)**,KP.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For **NUMMRG (p. 1295)**,ELEM, elements must be identical in all aspects, including the direction of the element coordinate system.

For certain solid and shell elements, the program interprets coincident faces as internal and eliminate them. To prevent this from occurring, shrink the entities by a very small factor to delineate coincident items (**/SHRINK (p. 1794)**, 0.0001) and no internal nodes, lines, areas or elements will be eliminated.

When working with solid models, you may have better success with the gluing operations (**AGLUE (p. 108)**, **LGLUE (p. 1047)**, **VGLUE (p. 2130)**). Please read the following information when attempting to merge solid model entities.

Gluing Operations vs. Merging Operations

Adjacent, touching regions can be joined by gluing them (**AGLUE (p. 108)**, **LGLUE (p. 1047)**, **VGLUE (p. 2130)**) or by merging coincident keypoints (**NUMMRG (p. 1295)**,KP, which also causes merging of identical lines, areas, and volumes). In many situations, either approach will work just fine. Some factors, however, may lead to a preference for one method over the other.

Geometric Configuration

Gluing is possible regardless of the initial alignment or offset of the input entities. Keypoint merging is possible only if each keypoint on one side of the face to be joined is matched by a coincident keypoint on the other side. This is commonly the case after a symmetry reflection (**ARSYM (p. 184)** or **VSYMM (p. 2169)**) or a copy (**AGEN (p. 107)** or **VGEN (p. 2116)**), especially for a model built entirely in Mechanical APDL rather than imported from a CAD system. When the geometry is extremely precise, and the configuration is correct for keypoint merging, **NUMMRG (p. 1295)** is more efficient and robust than **AGLUE (p. 108)** or **VGLUE (p. 2130)**.

Model Accuracy

As with all boolean operations, gluing requires that the input entities meet the current boolean tolerance (BTOL). Otherwise, **AGLUE (p. 108)** or **VGLUE (p. 2130)** may fail. In such cases, relaxing the tolerance may allow the glue to complete. An advantage of gluing is that it is unlikely to degrade the accuracy of a geometric model. Keypoint merging can operate on almost any combination of entities (although you may have to override the default tolerances on **NUMMRG (p. 1295)**). However, it can also introduce or increase accuracy flaws, making later boolean operations less likely to succeed. If the input tolerances are too large, **NUMMRG (p. 1295)** can collapse out small lines, areas, or volumes you intended to keep, possibly rendering the model unusable.

Mesh Status

As with all boolean operations, gluing requires that the input entities be unmeshed. Keypoint merging is effective for meshed models under the right conditions. More information on keypoint merging follows.

Merging Solid Model Entities:

When merging solid model entities (*Label* = KP or ALL), keypoint locations are used as the basis for merging. Once keypoints are merged, any higher order solid model entities (lines, areas, and volumes), regardless of their select status or attachment to the merged keypoints, are considered for merging.

Keypoints that are attached to lines will be merged only if:

- ΔX, ΔY, and ΔZ are each less than *TOLER*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where,

ΔX is the X component of the distance between keypoints,

ΔY is the Y component of the distance between keypoints, and

ΔZ is the Z component of the distance between keypoints;

and

- $\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$ is less than 1E-5 times the length of the longest line attached to those keypoints (internal relative solid model tolerance), or $\sqrt{\Delta X^2 + \Delta Y^2 + \Delta Z^2}$ is less than *GTOLER* (global solid model tolerance) if specified.

The *TOLER* field is a *consideration* tolerance. If a keypoint is within *TOLER* of another keypoint, then those two keypoints are *candidates* to be merged. If, when "moving" the higher numbered keypoint, the distance exceeds the internal relative solid model tolerance, or the global solid model tolerance (*GTOLER*) if specified, the keypoints will *not* be merged. Lines, areas, and volumes are considered for merging in a similar manner.

The internal relative solid model tolerance should be overridden by the global solid model tolerance (*GTOLER*) only when absolutely necessary. *GTOLER* is an *absolute* tolerance; if specified, relative lengths of lines in the model will no longer be considered in the merge operation. If *GTOLER* is too large, you can "merge-out" portions of your model accidently, effectively defeaturing the model. If using *GTOLER*, it is good practice so first save the database before issuing **NUMMRG (p. 1295)** (as undesired merges of solid model entities could occur).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Merge Nodes

Main Menu>Preprocessor>Modeling>Delete>Pre-tens Elemnts

Main Menu>Preprocessor>Numbering Ctrls>Merge Items

NUMOFF, *Label*, *VALUE*, *KREF*

Adds a number offset to defined items.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Apply offset number to one of the following sets of items:

NODE

Nodes

ELEM

Elements

KP

Keypoints

LINE

Lines

AREA

Areas

VOLU

Volumes

MAT

Materials

TYPE

Element types

REAL

Real constants

CP

Coupled sets

SECN

Section numbers

CE

Constraint equations

CSYS

Coordinate systems

VALUE

Offset number value (cannot be negative)

KREF

Attribute reference key:

0 – Add number offset to defined items only (default)

1 – Add number offset to *all* attribute references (includes undefined items)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Useful for offsetting current model data to prevent overlap if another model is read in. **CDWRITE (p. 293)** automatically writes the appropriate **NUMOFF (p. 1299)** commands followed by the model data to `File.cdb`. When the file is read, therefore, any model already existing in the database is offset before the model data on the file is read.

Offsetting material numbers with this command (**NUMOFF (p. 1299)**,MAT) does *not* update the material number referenced by either of the following:

- A temperature-dependent convection or surface-to-surface radiation load (**SF (p. 1733)**, **SFE (p. 1761)**, **SFL (p. 1774)**, **SFA (p. 1739)**)
- Real constants for multi-material elements

A mismatch may therefore exist between the material definitions and the material numbers referenced.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Add Num Offset

NUMSTR, *Label*, *VALUE*

Establishes starting numbers for automatically numbered items.

PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Apply starting number to one of the following sets of items:

NODE

Node numbers. *Value* defaults (and is continually reset) to 1 + maximum node number in model. Cannot be reset lower.

ELEM

Element numbers. *Value* defaults (and is continually reset) to 1 + maximum element number in model. Cannot be reset lower.

KP

Keypoint numbers. *Value* defaults to 1. Only undefined numbers are used. Existing keypoints are not overwritten.

LINE

Line numbers. *Value* defaults to 1. Only undefined numbers are used. Existing lines are not overwritten.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AREA

Area numbers. *Value* defaults to 1. Only undefined numbers are used. Existing areas are not overwritten.

VOLU

Volume numbers. *Value* defaults to 1. Only undefined numbers are used. Existing volumes are not overwritten.

DEFA

Default. Returns all starting numbers to their default values.

VALUE

Starting number value.

Notes

Establishes starting numbers for various items that may have numbers automatically assigned (such as element numbers with the **EGEN (p. 603)** command, and node and solid model entity numbers with the mesh like **AMESH (p. 121)**, **VMESH (p. 2140)**, etc.. Use **NUMSTR (p. 1301)**,STAT to display settings. Use **NUMSTR (p. 1301)**,DEFA to reset all specifications back to defaults. Defaults may be lowered by deleting and compressing items (that is, **NDELE (p. 1211)** and **NUMCMP (p. 1291)**,NODE for nodes, etc.).

Note:

A mesh clear operation (**VCLEAR (p. 2076)**, **ACLEAR (p. 93)**, **LCLEAR (p. 1024)**, and **KCLEAR (p. 975)**) automatically sets starting node and element numbers to the highest unused numbers. If a specific starting node or element number is desired, issue **NUMSTR (p. 1301)** after the clear operation.

Menu Paths

Main Menu>Preprocessor>Numbering Ctrls>Reset Start Num

Main Menu>Preprocessor>Numbering Ctrls>Set Start Number

Main Menu>Preprocessor>Numbering Ctrls>Start Num Status

NUMVAR, *NV*

Specifies the number of variables allowed in POST26.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV

Allow storage for *NV* variables. 200 maximum are allowed. Defaults to 10. TIME (variable 1) should also be included in this number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Allow storage for 10 variables.

Notes

Specifies the number of variables allowed for data read from the results file and for data resulting from an operation (if any). For efficiency, *NV* should not be larger than necessary. *NV* cannot be changed after data storage begins.

Menu Paths

Main Menu>TimeHist Postpro>Settings>File

NUSORT

Restores original order for nodal data.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command restores the nodal data to its original order (sorted in ascending node number sequence) after an **NSORT (p. 1283)** command. Changing the selected nodal set (**NSEL (p. 1266)**) also restores the original nodal order.

Menu Paths

Main Menu>General Postproc>List Results>Sorted Listing>Unsort Nodes

---

NWPAVE, *N1, N2, N3, N4, N5, N6, N7, N8, N9*

Moves the working plane origin to the average location of nodes.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

N1, N2, N3, . . . , N9

Nodes used in calculation of the average. At least one must be defined. If *N1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Averaging is based on the active coordinate system.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>WorkPlane>Offset WP to>Nodes

NWPLAN, *WN*, *NORIG*, *NXAX*, *NPLAN*

Defines the working plane using three nodes.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

NORIG

Node number defining the origin of the working plane coordinate system. If *NORIG* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NXAX

Node number defining the x-axis orientation (defaults to the x-axis being parallel to the global X-axis; or if the normal to the working plane is parallel to the global X-axis, then defaults to being parallel to the global Y-axis).

NPLAN

Node number defining the working plane (the normal defaults to the present display view (**/VIEW (p. 2131)**) of window *WN*).

Notes

Defines a working plane to assist in picking operations using three nodes as an alternate to the **WPLANE (p. 2180)** command. The three nodes also define the working plane coordinate system. A minimum of one node (at the working plane origin) is required. Immediate mode may also be active. See the **WPSTYL (p. 2182)** command to set the style of the working plane display.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node

Utility Menu>WorkPlane>Align WP with>Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NWRITE, *Fname*, *Ext*, --, *KAPPND*

Writes nodes to a file.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to NODE if *Fname* is blank.

--

Unused field.

KAPPND

Append key:

0

Rewind file before the write operation.

1

Append data to the end of the existing file.

Notes

Writes selected nodes (**NSEL (p. 1266)**]) to a file. The write operation is not necessary in a standard Mechanical APDL run but is provided as a convenience to those who want coded node file.

Data are written in a coded format. The format used is (I9, 6G21.13E3) to write out *NODE*,*X*,*Y*,*Z*,*THXY*,*THYZ*,*THZX*. If the last number is zero (*THZX* = 0), or the last set of numbers are zero, they are not written but are left blank. Therefore, use a formatted read to process this file.

Coordinate values are in the global Cartesian system.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Write Node File

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

O Commands

OCDATA, *VAL1*, *VAL2*, *VAL3*, . . . , *VAL14*

Defines an ocean load using non-table data.

SOLUTION (p. 38): Ocean (p. 49)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1*, *VAL2*, *VAL3*, . . . , *VAL14

Values describing the basic ocean load or a wave condition.

Notes

The **OCDATA (p. 1307)** command specifies non-table data that defines the ocean load, such as the depth of the ocean to the mud line, the ratio of added mass over added mass for a circular cross section, or the wave type to apply. The terms *VAL1*, *VAL2*, etc. are specialized according to the input set required for the given ocean load.

The program interprets the data input via the **OCDATA (p. 1307)** command within the context of the most recently issued **OCTYPE (p. 1320)** command.

Input values in the order indicated.

This command is also valid in PREP7.

You can define the following ocean data types:

Ocean Data Type: Basic (OCTYPE,BASIC)

Ocean Data Type: Wave (OCTYPE,WAVE)

Ocean Zone Type: Component, Z-Level, or Pipe (OCZONE,COMP/ZLOC/PIP)

Ocean Data Type: Basic (OCTYPE (p. 1320),BASIC)

For a better understanding of how to set up a basic ocean type, see Figure 7: Basic Ocean Data Type Components (p. 1309).

VAL1

DEPTH -- The depth of the ocean (that is, the distance between the mean sea level and the mud line). The water surface is assumed to be level in the XY plane, with Z being positive upwards. This value is required and must be positive.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL2

MATOC -- The material number of the ocean. This value is required and is used to input the required density. It is also used to input the viscosity if the Reynolds number is used (**OCTABLE (p. 1316)**).

VAL3

KFLOOD -- The inside-outside fluid-interaction key:

0 -- The density and pressure of fluid inside and outside of the pipe element (PIPE288 or PIPE289) are independent of each other. This behavior is the default.

1 -- The density and pressure of fluid inside of the pipe element (PIPE288 or PIPE289) are set to equal the values outside of the pipe element.

For beam subtype CTUBE and HREC used with BEAM188 or BEAM189 and ocean loading, *KFLOOD* is always set to 1.

VAL4

Cay -- The ratio of added mass of the external fluid over the mass of the fluid displaced by the element cross section in the y direction (normal). The added mass represents the mass of the external fluid (ocean water) that moves with the pipe, beam, or link element when the element moves in the element y direction during a dynamic analysis.

If no value is specified, and the coefficient of inertia *CMy* is not specified (**OCTABLE (p. 1316)**), both values default to 0.0.

If no value is specified, but *CMy is* specified, this value defaults to *Cay* = *CMy* - 1.0.

If this value *should* be 0.0, enter 0.0.

VAL5

Caz -- The ratio of added mass of the external fluid over the mass of a cross section in the element z direction (normal). The added mass represents the mass of the external fluid (ocean water) that moves with the pipe, beam, or link element when the element moves in the element z direction during a dynamic analysis.

If no value is specified, and *Cay* is specified, this value defaults to *Cay*.

If no value is specified, and the coefficient of inertia *CMz* is not specified (**OCTABLE (p. 1316)**), both values default to 0.0.

If no value is specified, but *CMz is* specified, this value defaults to *Cay* = *CMz* - 1.0.

If this value *should* be 0.0, enter 0.0.

VAL6

Cb -- The ratio of buoyancy force used over buoyancy force based on the outside diameter and water density. Accept the default value in most cases. Adjust this option only when you must account for additional hardware (such as a control valve) attached to the pipe exterior. A non-default value

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may lead to small non-physical inconsistencies; testing is therefore recommended for non-default values.

If no value is specified, this value defaults to 1.0.

If this value should be 0.0 (useful when troubleshooting your input), enter 0.0.

VAL7

Zmsl -- A vertical offset from the global origin to the mean sea level. The default value is zero (meaning that the origin is located at the mean sea level).

Two example cases for *Zmsl* are:

- A structure with its origin on the sea floor (*Zmsl* = *DEPTH*).
- A tidal change (*tc*) above the mean sea level (*Zmsl* = *tc*, and *DEPTH* becomes *DEPTH* + *tc*)

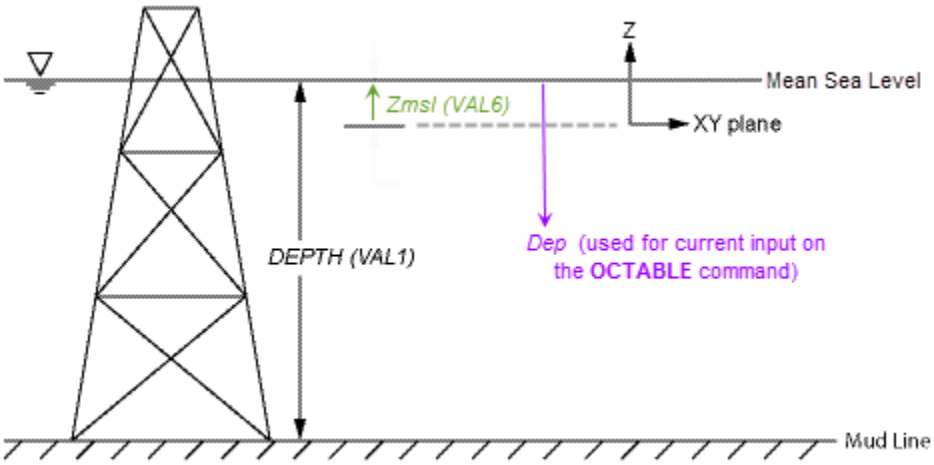
VAL8

Ktable -- The dependency of *VAL1* on the **OCTABLE (p. 1316)** command:

Z (or 1) -- Values on the **OCTABLE (p. 1316)** command depend on the Z levels (default).

RE (or 2) -- Values on the **OCTABLE (p. 1316)** command depend on the Reynolds number.

Figure 7: Basic Ocean Data Type Components



Ocean Data Type: Wave (OCTYPE (p. 1320),WAVE)

VAL1

KWAVE -- The incident wave type:

0 *or* AIRY -- Small amplitude Airy wave without modifications (default).

1 *or* WHEELER -- Small amplitude wave with Wheeler empirical modification of depth decay function.

2 *or* STOKES-- Stokes fifth-order wave.

3 *or* STREAMFUNCTION -- Stream function wave.

5 *or* RANDOM -- Random (but repeatable) combination of linear Airy wave components.

6 *or* SHELLNEWWAVE -- Shell new wave.

7 *or* CONSTRAINED -- Constrained new wave.

8 *or* DIFFRACTED -- Diffracted wave (using imported hydrodynamic data)

101+ -- API for computing particle velocities and accelerations due to waves and current:

101 through 200 -- Data preprocessed (via *KWAVE* = 0 logic).

201+ -- Data not preprocessed.

For more information, see the description of the `userPartVelAcc` subroutine in the *Programmer's Reference*.

VAL2

THETA -- Angle of the wave direction θ from the global Cartesian X axis toward the global Cartesian Y axis (in degrees).

VAL3

WAVELOC (valid when *KWAVE* = 0 through 3, and 101+) -- The wave location type:

0 -- Waves act on elements at their actual locations (default).

1 -- Elements are assumed to be at wave peak.

2 -- Upward vertical wave velocity acts on elements.

3 -- Downward vertical wave velocity acts on elements.

4 -- Elements are assumed to be at wave trough.

SPECTRUM (valid when *KWAVE* = 5 through 7) -- The wave spectrum type:

0 -- Pierson-Moskowitz (default).

1 -- JONSWAP.

2 -- User-defined spectrum.

VAL4

KCRC -- The wave-current interaction key.

Adjustments to the current profile are available via the *KCRC* constant of the water motion table. Typically, these options are used only when the wave amplitude is large relative to the water depth, such that significant wave-current interaction exists.

0 -- Use the current profile (as input) for wave locations below the mean water level, and the top current profile value for wave locations above the mean water level (default).

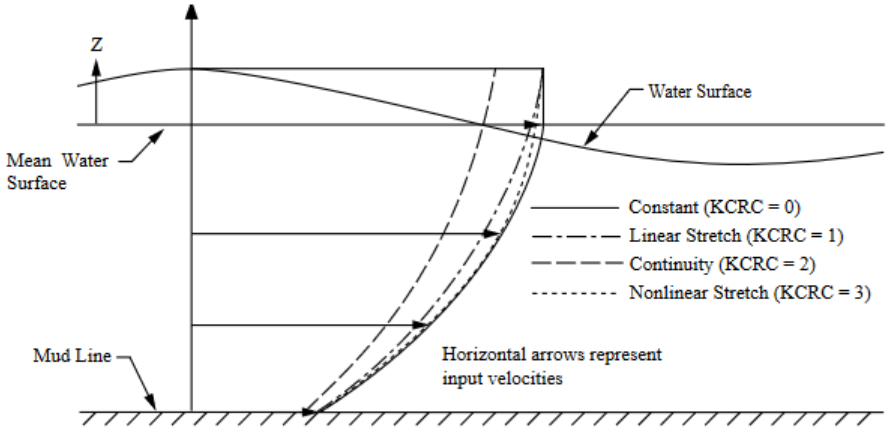
1 -- Linearly stretch or compress the current profile from the mud line to the top of the wave.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 -- Similar to *KCRC* = 1, but also adjusts the current profile horizontally such that total flow continuity is maintained with the input profile. All current directions *Th*(j) must be identical.

The following option is valid only when KWAVE = 5 through 7:

3 -- Nonlinear stretch or compress the current profile, as recommended in *API RP 2A Codes of Practice for Designing and Constructing Fixed Offshore Platforms*.



VAL5

KMF -- The MacCamy-Fuchs adjustment key, typically used only for larger-diameter pipes in relatively shallow water:

0 -- Do not apply the adjustment (default).

1 -- Apply the adjustment (valid only when *KWAVE* = 0 or 1).

VAL6

PRKEY -- The wavelength wave-printout key:

0 -- No extra printout (default).

1 -- Include the extra printout.

2 -- Print wave component details (valid only when *KWAVE* = 5 through 7).

The following input values are valid only when *KWAVE* = 5 through 7:

VAL7

APC -- Activate apparent period calculation when a wave is superimposed upon a current:

0 -- Not activated (default).

1 -- Activated.

VAL8

DSA -- Stretching depth factor:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stretching is performed between a distance of *DSA* * *Hs* below the mean water level (MWL) and the water surface, where *Hs* is the significant wave height measured from the MWL. No stretching occurs outside this range, or if the wave surface is below the MWL. If *DSA* * *Hs* is negative, stretching is performed between that level above the MWL and the water surface. The default *DSA* value is 0.5.

VAL9

DELTA -- Delta stretching parameter (0.0 ≤ *DELTA* ≤ 1.0):

A value of 0.0 corresponds to Wheeler stretching under wave crests, 1.0 corresponds to linear extrapolation of kinematics at mean water level to crest. (Default = 0.3.) If zero is required, specify a small positive number (0.01 or less) instead.

VAL10

Wave kinematics factor or wave spreading angle:

KINE (*KWAVE* = 5 or 7) -- Wave kinematics factor (0.0 < *KINE* ≤ 1.0). The factor is used to account for wave spreading by modifying the horizontal wave velocities and accelerations. A value of 1.0 corresponds to uni-directional wave with no spreading. (Default = 1.0, no spreading.)

SPANGLE (*KWAVE* = 6) -- Wave spreading angle in degrees (0.0 ≤ *SPANGLE* ≤ 40.0). The angle is used to compute a wave spreading factor to modify the horizontal wave kinematics for nearly unidirectional seas. *SPANGLE* = 0.0 corresponds to no spreading. (Default = 0.0, no spreading.)

VAL11

Random seed value for phase angle generation, or wave crest amplitude value:

SEED (*KWAVE* = 5) -- Initial seed for random phase angle generation. (Default = 1.)

AMPMAX (*KWAVE* = 6) -- Maximum wave crest amplitude (distance between the mean water level and maximum wave crest).

AMPCONST (*KWAVE* = 7) -- Constrained wave crest amplitude (distance between the mean water level and wave crest).

The following input values are valid only when *KWAVE* = 6 or 7:

VAL12

TOFF -- Time offset at which the maximum wave crest will occur. (Default = 0.0.)

VAL13

ROFF -- Position offset along the wave direction where the maximum wave crest will occur. (Default = 0.0.)

VAL14

EVOLVING (*KWAVE* = 6) -- Activate evolving wave:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 -- Not activated (default).

1 -- Activated.

SEED (*KWAVE* = 7) -- Initial seed for random phase angle generation. (Default = 1.)

Usage Hints for *KWAVE* = 5 Through 7

Recommendations given in *API RP 2A Codes of Practice for Designing and Constructing Fixed Offshore Platforms* can be enforced via the following settings (valid only when *KWAVE* = 5 through 7):

- Set *KCRC* = 3 (*VAL4*) to apply nonlinear current stretching.
- Set *APC* = 1 (*VAL7*) to enable automatic computation of apparent wave period when a wave is superimposed upon a current.
- If *KWAVE* = 5 or 7, specify wave kinematics factor *KINE* (*VAL10*) to modify the horizontal velocities and accelerations due to wave spreading. The factor can be estimated from measurement or hindcast directional spectral wave data as $\sqrt{(n+1)/(n+2)}$, where n is the exponent in the cosine wave spreading function at the spectral peak frequency.
- If *KWAVE* = 6, specify wave spreading angle *SPANGLE* (*VAL10*) to modify the horizontal velocities and accelerations due to wave spreading. This parameter differs from the wave kinematics factor indicated in the API codes, but the principle is similar nevertheless.

For *VAL11* (when *KWAVE* = 6 or 7), if no maximum or constrained wave crest amplitude (*AMPCONST* or *AMPMAX*) is defined, the value is calculated based on the significant wave height obtained from the spectrum data (**OCTABLE (p. 1316)**). For further information, see Hydrodynamic Loads in the *Mechanical APDL Theory Reference*.

For *VAL14*, the Shell new wave model (*KWAVE* = 6) is evolving (or *dispersive*) in that the crest height varies with time. By default, however, the wave is "frozen" so that it can be stepped through in a quasi-static manner.

When using waves in a superelement generation run (**ANTYPE (p. 162)**,SUBSTR), consider whether you should take the ocean level into account (*SeOcLvL* on the **SEOPT (p. 1715)** command).

Ocean Zone Type: Component, Z-Level, or Pipe (**OCZONE (p. 1322)**,COMP/ZLOC/PIP)

An ocean zone is a *local* space where you can override global ocean-loading parameters. The following arguments specifying the ocean zone values are described in more detail under "Ocean Data Type: Basic (OCTYPE,BASIC)" (p. 1307).

VAL1

KFLOOD -- The inside-outside fluid-interaction key.

VAL2

Cay -- The ratio of added mass of the external fluid over the mass of a cross section in the element y direction (normal).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL3

Caz -- The ratio of added mass of the external fluid over the mass of a cross section in the element z direction (normal).

VAL4

Cb -- The ratio of buoyancy force used over buoyancy force based on the outside diameter and water density.

Ocean Zone Type Usage Notes

Ocean zone values specified via the **OCDATA (p. 1307)** command override global ocean-loading parameters.

Arguments not specified default to the global values specified for the basic ocean type. Therefore, the relationship between *Ca* and *CM* values (*Ca* = *CM* - 1.0) is not applied to ocean zones.

For a pipe-type ocean zone (**OCZONE (p. 1322)**,PIP), *KFLOOD* is the only valid option.

Menu Paths

This command cannot be accessed from a menu.

OCDELETE,*DataType*,*ZoneName*

Deletes a previously defined ocean load.

SOLUTION (p. 38): Ocean (p. 49)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DataType

Ocean data type to delete. Valid values are BASIC, CURRENT, WAVE, ZONE, and ALL.

For *DataType* = ALL, *all* defined ocean loads are deleted.

ZoneName

The name of the ocean zone to delete. If no name is specified, *all* defined ocean zones are deleted. Valid only when *DataType* = ZONE.

Notes

The **OCDELETE (p. 1314)** command deletes previously specified ocean data from the database.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OCLIST,*DataType*,*ZoneName*

Summarizes all currently defined ocean loads.

SOLUTION (p. 38): Ocean (p. 49)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DataType

Ocean data type to list. Valid values are BASIC, CURRENT, WAVE, ZONE, and ALL.

For *DataType* = ALL, *all* defined ocean loads are listed.

ZoneName

The name of an ocean zone to list. If no name is specified, *all* defined ocean zones are listed. Valid only when *DataType* = ZONE.

Notes

The **OCLIST (p. 1315)** command summarizes the ocean properties for all defined ocean loads in the current session.

When this command follows the **SOLVE (p. 1822)** command, certain waves types also list the calculated wave length.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

OCREAD,*Fname*,*Ext*,--,*Option*

Reads externally defined ocean data.

SOLUTION (p. 38): Ocean (p. 49)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

External ocean data file name (excluding the filename extension) and directory path containing the file. For more information, see the Notes (p. 1316) section.

Ext

Filename extension (limited to eight characters).

--

Reserved field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option

Integer value passed to the `userOceanRead` subroutine (as *`iOption`*) for user-defined waves. This value does not apply to the diffracted wave type.

Notes

The **OCREAD (p. 1315)** command imports ocean data that has been defined externally (for example, via the Hydrodynamic Diffraction System (AQWA)).

The command operates on the ocean load ID specified via the most recently issued **OCTYPE (p. 1320)** command. Issue a separate **OCREAD (p. 1315)** command for each ocean load that you want to read into the program.

`Fname` is limited to 248 characters, including the directory path. If *`Fname`* does not include a directory path, the program searches for the specified file in the current working directory. An unspecified *`Fname`* defaults to `Jobname`.

For the diffracted wave type (*KWAVE* = 8 on the **OCDATA (p. 1307)** command), you must issue an **OCREAD (p. 1315)** command for the ocean wave ID in order to import the hydrodynamic data from the hydrodynamic analysis.

For more information, see Applying Ocean Loading from a Hydrodynamic Analysis in the *Advanced Analysis Guide*.

To learn more about creating user-defined waves, see Subroutine userPanelHydFor (Calculating Panel Loads Caused by Ocean Loading) in the *Programmer's Reference*.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

OCTABLE, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*

Defines an ocean load using table data.

`SOLUTION` (p. 38): Ocean (p. 49)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1*, *VAL2*, *VAL3*, . . . , *VAL6

Values describing the basic ocean load, a current condition, or a wave condition.

Notes

The **OCTABLE (p. 1316)** specifies table data that defines the ocean load. The terms *VAL1*, *VAL2*, etc. are specialized according to the input set required for the given ocean load.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The program interprets the data input via the **OCTABLE (p. 1316)** command within the context of the most recently issued **OCTYPE (p. 1320)** command.

There is no limit to the number of data input.

Input values in the order indicated.

This command is also valid in PREP7.

You can define the following ocean data types:

Ocean Data Type: Basic (OCTYPE,BASIC)

Ocean Data Type: Current (OCTYPE,CURR)

Ocean Data Type: Wave (OCTYPE,WAVE)

Ocean Zone Type: Component or Z-Level (OCZONE,COMP/ZLOC)

Ocean Data Type: Basic (**OCTYPE (p. 1320)**,BASIC)

Basic ocean data to provide in the value fields:

IndVar, --, *CDy*, *CDz*, *CT*, *CMy*, *CMz*

where

IndVar = Independent variable for the table inputs. This value is dependent on the *Ktable* value specified via the **OCDATA (p. 1307)** command. If *Ktable* = Z, enter this value in *descending* order on each **OCTABLE (p. 1316)** command. If *Ktable* = RE, enter this value field in *ascending* order.

-- = Reserved.

CDy = Drag coefficient in the element y direction (normal).

CDz = Drag coefficient in the element z direction (normal). This value defaults to *CDy*.

CT = Drag coefficient in the element x direction (tangential).

CMy = Coefficient of inertia in the element y direction. If no value is specified, and *Cay* is specified, this value defaults to *Cay* + 1.0. If neither this value nor *Cay* is specified, both values default to 0.0.

CMz = Coefficent of inertia in the element z direction. If no value is specified, and *CMy* is specified on the same **OCTABLE (p. 1316)** command, this value defaults to *CMy*. If neither this value nor *CMy* is specified, and *Caz* is specified, this value defaults to *Caz* + 1.0. If neither this value nor *Caz* is specified, both values default to 0.0.

Ocean Data Type: Current (**OCTYPE (p. 1320)**,CURR)

Current data to provide in the value fields:

Dep, *W*, *Th*, *Te*

where

Dep = Depth of the drift current being input. Input these values in ascending order from one command to the next.

- If the current is constant, only one **OCTABLE (p. 1316)** command is necessary and *Dep* is not required.
- For waves (p. 1318), the current profile is stretched or compressed linearly up to 10 percent.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The first *Dep* value (representing the mean sea level) must be zero. The last *Dep* value (representing the mud line) must be equal to the *DEPTH* value input on the **OCDATA (p. 1307)** command.
- The Cartesian Z values used to locate nodes, etc. decrease as one moves from the ocean surface to the sea floor, but the *Dep* values increase. See Figure 7: Basic Ocean Data Type Components (p. 1309).
- *Dep* is not affected by changes to *Zmsl* on the **OCDATA (p. 1307)** command, as that value simply relocates the origin.

W = Velocity of the drift current at this location.

Th = Angle of the drift current from the global Cartesian X axis toward the global Cartesian Y axis (in degrees) at this location.

Te = Temperature at this location.

Ocean Data Type: Wave (OCTYPE (p. 1320),WAVE)

When specifying an ocean wave type, issue the **OCTABLE (p. 1316)** command to input either wave location data (p. 1318) or wave spectrum data (p. 1319).

Wave Location Input Data

Wave location data to provide in the value fields (valid only when *KWAVE* = 0 through 3, or 8, on the **OCDATA (p. 1307)** command):

H, *T*, *Ps*, *L*, *NORDER*, *KPRCO*

where

H = Wave height (peak-to-trough).

T = Wave period.

Ps = Phase shift (in degrees)

L = Wavelength. An optional value used only when *KWAVE* = 0 or 1 (and ignored for all other *KWAVE* types).

NORDER = Order used by stream function wave theory (*KWAVE* = 3). This value is optional.

KPRCO = Key for printing (1) or not printing (0 and default) the calculated dimensionless coefficients of the stream function wave theory (*KWAVE* = 3). This value is optional.

Hints for Wave Location Input:

- The **TIME (p. 2000)** command is not used, except perhaps to identify the load case.
- The phase shift (*Ps*) determines the wave position (that is, the point at which the load is to be applied).
- When using the Stokes fifth-order (*KWAVE* = 2) or stream function (*KWAVE* = 3) wave type, issue only one **OCTABLE (p. 1316)** command.
- The valid range of the order of the stream function (*NORDER*) is 3 through 50. If no value is specified, the program determines a value automatically.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When using the diffracted wave type (*KWAVE* = 8), an **OCREAD (p. 1315)** command is also required to read in the hydrodynamic data from the hydrodynamic analysis.

Wave Spectrum Input Data

Wave spectrum data to provide in the value fields (valid only when *KWAVE* = 5 through 7 on the **OCDATA (p. 1307)** command):

***SPECTRUM* = 0** (Pierson-Moskowitz spectrum)

HS, *TP*, *NWC*

where

HS = Significant wave height of the spectrum.

TP = Peak period for the spectrum.

NWC = Number of wave components ($1 \le NWC \le 1000$) to model the spectrum. (Default = 50.)

***SPECTRUM* = 1** (JONSWAP spectrum)

HS, *TP*, *GAMMA*, *NWC*

where

HS = Significant wave height of the spectrum.

TP = Peak period for the spectrum.

GAMMA = Peak enhancement factor for the spectrum. (Default = 3.3.)

NWC = Number of wave components ($1 \le NWC \le 1000$) to model the spectrum. (Default = 50.)

***SPECTRUM* = 2** (User-defined spectrum)

w, *s*, *NWC*

w = Angular frequency (rad/s).

s = Spectral energy density (Length^2 / (rad/s))

NWC = Number of wave components ($1 \le NWC \le 1000$) to model the spectrum. (Default = 50.)

Hints for Wave Spectrum Input:

- When defining a Pierson-Moskowitz or JONSWAP spectrum (*SPECTRUM* = 0 or 1, respectively, on the **OCDATA (p. 1307)** command), issue only one **OCTABLE (p. 1316)** command.

- When defining a Pierson-Moskowitz or JONSWAP spectrum for Shell new wave (*KWAVE* = 6 on the **OCDATA (p. 1307)** command), *HS* is calculated from the maximum wave crest amplitude (*AMPMAX* on the **OCDATA (p. 1307)** command) if no value is specified. For further information, see Hydrodynamic Loads in the *Mechanical APDL Theory Reference*.

- For a user-defined spectrum (*SPECTRUM* = 2 on the **OCDATA (p. 1307)** command), issue an **OCTABLE (p. 1316)** command for *each* frequency data point defining the spectrum. Specify the frequency data in ascending order. The number of wave components (*NWC*) is required on the *first* **OCTABLE (p. 1316)** command only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ocean Zone Type: Component or Z-Level (OCZONE (p. 1322),COMP/ZLOC)

An ocean zone is a *local* space where you can override global ocean-loading parameters.

Ocean zone data to provide in the value fields:

Z, --, *CDy*, *CDz*, *CT*, *CMy*, *CMz*, *Mbio*, *Tbio*

where

Z = Z level for the coefficients specified on this command.

-- = Reserved.

CDy = Drag coefficient in the element y direction (normal).

CDz = Drag coefficient in the element z direction (normal). This value defaults to *CDy*.

CT = Drag coefficient in the element x direction (tangential).

CMy = Coefficient of inertia in the element y direction.

CMz = Coefficient of inertia in the element z direction. This value defaults to *CMy*.

Mbio = Material ID of biofouling.

Tbio = Thickness of biofouling.

Ocean Zone Type Usage Notes

Ocean zone values specified via the **OCTABLE (p. 1316)** command override global ocean-loading parameters.

Arguments not specified default to the global values specified for the basic ocean type (**OCTYPE (p. 1320)**,BASIC). Therefore, the relationship between *Ca* and *CM* values (*Ca* = *CM* - 1.0) is not applied to ocean zones.

The **OCTABLE (p. 1316)** command is not valid for a pipe-type ocean zone (**OCZONE (p. 1322)**,PIP).

Menu Paths

This command cannot be accessed from a menu.

OCTYPE,*DataType*,*Name*

Specifies the type of ocean load data to follow.

SOLUTION (p. 38): Ocean (p. 49)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DataType

The type of ocean data to be input following this command:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BASIC

The basic ocean load, required for any ocean loading.

CURR

An optional drift current.

WAVE

An optional ocean wave state.

Specify basic, current, or wave input data via the **OCDATA (p. 1307)** and **OCTABLE (p. 1316)** commands. The example input fragment listed in the Notes (p. 1701) section shows how to use the ocean load data types.

Name

An eight-character name for the ocean load. An ocean name can consist of letters and numbers, but cannot contain punctuation, special characters, or spaces.

Notes

The **OCTYPE (p. 1320)** command specifies the type of ocean load data to follow (basic, current, or wave). Issue this command *before* defining your ocean load data (**OCDATA (p. 1307)** and **OCTABLE (p. 1316)**).

Ocean loading applies only to current-technology pipe (PIPE288 and PIPE289), surface (SURF154), link (LINK180) and beam (BEAM188 and BEAM189) elements.

An ocean current or wave is accessible repeatedly. For example, it is not necessary to input an identical current table again just because the drag coefficients of the basic input table have changed.

The following example shows how you can use the basic (*DataType* = BASIC), current (*DataType* = CURR), and wave (*DataType* = WAVE) ocean data types within the context of a simple input file fragment:

```
Do=1.5                        ! outside diameter
th=0.1                        ! wall thickness
height=10                     ! wave height
CS=2                          ! surface current speed
Depth=100                     ! water depth
matwat=2                      ! material number id of the ocean
secpipe= 1                    ! section number of the pipe
!
sectype,secpipe,pipe,,pipetest
secdata,Do,th,16              ! 16 cells around circumference
!
octype,basic
ocdata,Depth,matwat,0,0,0,0   ! suppress added mass and buoyancy
octable,,,.5,.5,,2            ! CD = .5, CM = 2

octype,curr
octable,0.0,CS                ! input free surface current speed
octable,Depth,0.00            ! input ocean floor current speed of 0.0
!
octype,wave
ocdata,2                      ! request Stokes wave type
octable,height,8              ! wave period of 8 seconds

slist,all                     ! lists pipe section AND
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
!                                  mentions ocean loading
oclist,all                      ! lists details of ocean loading
```

Menu Paths

This command cannot be accessed from a menu.

OCZONE, *ZoneType*, *ZoneName*, *CompNameInt*, *CompNameExt*

Specifies the type of ocean zone data to follow.

SOLUTION (p. 38): Ocean (p. 49)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ZoneType

The type of ocean zone data to be input following this command:

COMP

Define by a component.

ZLOC

Define by Z levels.

PIP

Associate an internal pipe or pipes with an external pipe.

ZoneName

The ocean zone name. If no name is specified, the program assigns one.

CompNameInt

For *Zonetype* = COMP, the required name of a component.

For *Zonetype* = PIP, the required name of an internal pipe component.

CompNameExt

For *Zonetype* = PIP, the required name of an external pipe component.

Notes

The **OCZONE (p. 1322)** command specifies the type of ocean zone data to follow (component, Z-level, or internal pipes associated with an external pipe). An ocean zone is a *local* space where you can override global ocean-loading parameters.

Names specified for *ZoneName*, *CompNameInt*, and *CompNameExt* can consist of up to 32 alphanumeric characters. The name cannot contain punctuation, special characters, or spaces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

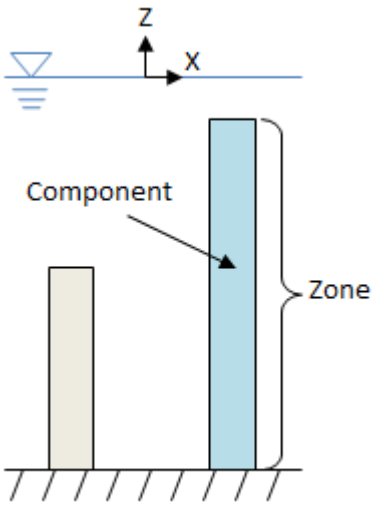
For *Zonetype* = COMP, the zone is defined by a component. Only the elements in the component are affected by the local parameters. A *partial* component can be defined as the zone via the Z input on the **OCTABLE (p. 1316)** command.

For *Zonetype* = ZLOC, the zone is defined by Z levels. Structural elements (such as BEAM188, BEAM189, PIPE288, PIPE289, and LINK180) in the Z levels are included in the zone.

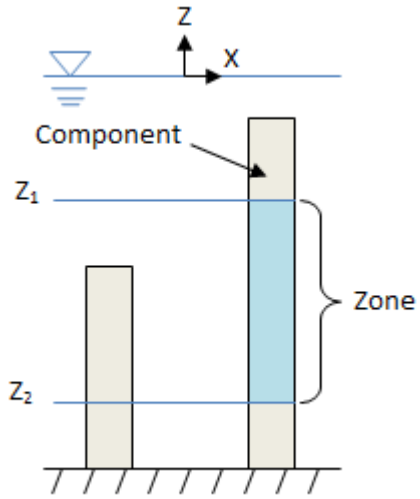
For *Zonetype* = PIP, the zone is prepared for a special configuration of pipes. It associates an internal pipe or pipes with an external pipe to remove the hydrodynamic effect on the internal pipe. Only hydrostatic pressure is applied on the internal pipe.

This command is also valid in PREP7.

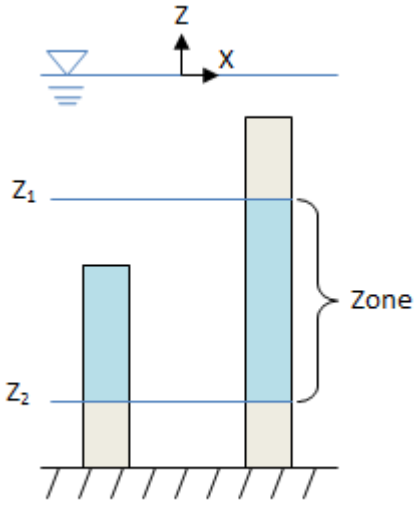
Figure 8: Ocean Zone Types (Specified via *ZoneType*)



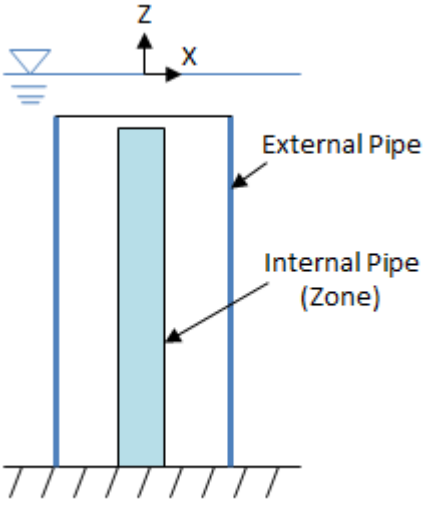
COMP



COMP with Z Inputs



ZLOC



PIP

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Issue this command before defining your ocean load data (**OCDATA (p. 1307)** or **OCTABLE (p. 1316)**). Define components before defining a component-type or a pipe-type zone (**OCZONE (p. 1322)**,COMP or **OCZONE (p. 1322)**,PIP, respectively).

Menu Paths

This command cannot be accessed from a menu.

OMEGA, *OMEGX*, *OMEGY*, *OMEGZ*

Specifies the rotational velocity of the structure.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

OMEGX*, *OMEGY*, *OMEGZ

Rotational velocity of the structure about the global Cartesian X, Y, and Z axes.

Notes

This command specifies the rotational velocity of the structure about each of the global Cartesian axes (right-hand rule). Rotational velocities may be defined in these analysis types:

- Static (**ANTYPE (p. 162)**,STATIC)
- Harmonic (**ANTYPE (p. 162)**,HARMIC) -- full, VT [1], Krylov [1], or mode-superposition [2]
- Transient (**ANTYPE (p. 162)**,TRANS) -- full or mode-superposition [2]
- Substructuring (**ANTYPE (p. 162)**,SUBSTR)
- Modal (**ANTYPE (p. 162)**,MODAL)

1. Loads for VT and Krylov methods are supported as long as they are not:
   - complex tabulated loads (constant or trapezoidal loads in tabulated form are supported)
   - used in conjunction with Rotordynamics (**CORIOLIS (p. 417)**,on).
2. In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

The command supports tabular boundary conditions (%*TABNAME_X*%, %*TABNAME_Y*%, and %*TABNAME_Z*%) for *OMEGA_X*, *OMEGA_Y*, and *OMEGA_Z* input values (***DIM (p. 530)**) for full transient and harmonic analyses.

Rotational velocities are combined with the element mass matrices to form a body-force load vector term. Units are radians/time. Related commands are **ACEL (p. 92)**, **CGLOC (p. 319)**, **CGOMGA (p. 320)**, **DCGOMG (p. 501)**, and **DOMEGA (p. 571)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Analysis Tools in the *Theory Reference* for more information.

If you have applied the Coriolis effect (**CORIOLIS (p. 417)**) using a stationary reference frame, this command takes the gyroscopic damping matrix into account for the elements listed in the **Stationary Reference Frame** heading in the notes section of the **CORIOLIS (p. 417)** command. The element axis must pass through the global Cartesian origin. The program verifies that the rotation vector axis is parallel to the axis of the element; if not, the gyroscopic effect is not applied. After issuing **OMEGA (p. 1324)** when the Coriolis or gyroscopic effect is present, a subsequently issued **CMOMEGA (p. 368)** command has no effect.

The load interpolation setting (**KBC (p. 970)**) applies to the rotational velocity, in particular the *OMG-SQRDKEY* option for quadratic interpolation.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>Global

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Inertia>Angular Veloc>Global

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>Global

Main Menu>Solution>Define Loads>Delete>Structural>Inertia>Angular Veloc>Global

---

OPERATE

Specifies "Operation data" as the subsequent status topic.

POST26 (p. 58): Status (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OPNCONTROL, *Lab*, *VALUE*, *NUMSTEP*

Sets decision parameter for automatically increasing the time step interval in a pure thermal analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

TEMP

Degree-of-freedom label used to base a decision for increasing the time step (substep) interval in a nonlinear or transient analysis. The only DOF label currently supported is TEMP.

OPENUPFACTOR

Key to set a multiplier for increasing the time step interval as specified in *VALUE* > 1.0 (up to 10.0). Valid only when **AUTOTS (p. 207)**,ON has been issued.

The multiplier is set by issuing **OPNCONTROL (p. 1326)**,OPENUPFACTOR,*VALUE* as follows:

- For a pure thermal analysis, if *VALUE* > 1.0, the OPENUPFACTOR is the minimum of 10.0,*VALUE*.

If the user does not specify the multiplier, the default is 3.0.

Generally, *VALUE* > 3.0 is not recommended. Note that in some rare cases this specification can be overwritten by internal heuristics in determining the new time step interval.

VALUE

A context sensitive value that depends on *Lab*:

- If *Lab* = TEMP, *VALUE* is used together with *NUMSTEP* in the algorithm for determining if the time step interval can be increased. The time step interval is increased if the maximum absolute value of the incremental solution is less than *VALUE* for the number of contiguous time steps specified by *NUMSTEP* (default *VALUE* = 0.1).
- If *Lab* = OPENUPFACTOR, *VALUE* is a multiplier (= 1.0 - 10.0) that can be specified for a pure thermal analysis as described above.

NUMSTEP

Valid only when *Lab* = TEMP. A value used together with *VALUE* in the algorithm for determining if the time step interval can be increased. The time step interval is increased if the maximum absolute value of the incremental solution at the specified TEMP label is less than *VALUE* for the number of contiguous time steps specified by *NUMSTEP* (default *NUMSTEP* = 3).

Notes

This command is available only for nonlinear static or full transient analyses. **OPNCONTROL (p. 1326)** enables an increase in the current time step size. It is analogous to the **CUTCONTROL (p. 445)** command, but with the opposite effect. **CUTCONTROL (p. 445)** reduces the step size for analyses experiencing

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

convergence difficulties while **OPNCONTROL (p. 1326)** increases the step size to speed up converging analyses.

The increase in the current time step size via **OPNCONTROL (p. 1326)** occurs when:

- a trigger mechanism is encountered and
- a multiplier (greater than 1.0) is set.

Different internal heuristics are used to automatically trigger the increase of the time step for different physics. However, for a pure thermal analysis, an additional trigger to increase the step size can be implemented by issuing **OPNCONTROL (p. 1326)**,TEMP.

The multiplier is set by issuing **OPNCONTROL (p. 1326)**,OPENUPFACTOR,*VALUE* (see details for different analysis types in the OPENUPFACTOR argument description above).

For linear full transient analysis, where the time step interval can be predominantly determined by the estimated modal frequency (number of solution points in a cycle in the dynamic system), the multiplier set via the OPENUPFACTOR argument may show no effect.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Open Control

Main Menu>Solution>Load Step Opts>Nonlinear>Open Control

OSRESULT,*Item*,*Comp*,*Freq*,*Cname*

Controls the selected result data written to the database.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Item to output to the database. See Table 208: OSRESULT - Item and Component Labels (p. 1328).

Comp

Component of *Item* to output to the database. See Table 208: OSRESULT - Item and Component Labels (p. 1328).

Freq

Frequency to output to the database.

n

Writes every *n*th and last substep of each load step.

-n

Writes up to *n* equally spaced substeps of each load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Writes every substep.

LAST

Writes the last substep of each load step (default).

Cname

The name of an element component (**CM (p. 356)**) defining the set of elements for which this specification is active. If not specified, the set is *all* elements.

Notes

OSRESULT (p. 1327) controls output to the results database for the selected result defined by the item and component combination. The command activates output of the selected result for the specified substeps and elements. Multiple commands for the same result are cumulative. No selected results are written to the database unless specified via **OSRESULT (p. 1327)**.

The saved selected quantities are accessible via standard postprocessing commands (**ANSOL (p. 149)**, **ETABLE (p. 687)**, **ESOL (p. 672)**, **PLESOL (p. 1401)**, **PLNSOL (p. 1425)**, **PRESOL (p. 1481)**, and **PRNSOL (p. 1505)**).

OSRESULT (p. 1327),ERASE deletes the existing output specifications.

OSRESULT (p. 1327),STATUS lists the current set of selected result specifications.

The output of selected results is valid for static (**ANTYPE (p. 162)**,STATIC) and transient (**ANTYPE (p. 162)**,TRANS) analysis types.

To select other results to output to the database, see **OUTRES (p. 1336)**. (Element quantities specified via **OUTRES (p. 1336)** can be redundant to those specified via **OSRESULT (p. 1327)**. Avoid specifying redundant quantities, as they are stored and processed separately.)

All **OSRESULT (p. 1327)** results are in the solution coordinate system.

This command is also valid in PREP7.

Table 208: OSRESULT - Item and Component Labels

Component Name Method		
Item	**Comp**	**Description**
ERASE	--	Erases all selected result output specifications
STATUS	--	Lists the current set of selected result output specifications
SVAR	1,2,3, …, *N*	State variable number
FLD	UF01, UF02, ... , UF09	User-defined field variables
S	X, Y, Z, XY, YZ, XZ, 1, 2, 3, INT, EQV	Component and derived (principal, intensity, equivalent) stresses
EPEL	X, Y, Z, XY, YZ, XZ, 1, 2, 3, INT, EQV	Component and derived (principal, intensity, equivalent) elastic strains

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Component Name Method		
Item	**Comp**	**Description**
EPPL	X, Y, Z, XY, YZ, XZ, 1, 2, 3, INT, EQV	Component and derived (principal, intensity, equivalent) plastic strains
EPCR	X, Y, Z, XY, YZ, XZ, 1, 2, 3, INT, EQV	Component and derived (principal, intensity, equivalent) creep strains
EPTH	X, Y, Z, XY, YZ, XZ, 1, 2, 3, INT, EQV	Component and derived (principal, intensity, equivalent) thermal strains

Example 10: Specifying and Retrieving Selected Result Output

```
/SOLU
OUTRES,ALL,NONE        ! Here we want only selected results output,
                       !   so we suppress solution-result data written
                       !   to the database via OUTRES.

! Issue OSRESULT to specify selected results.
OSRESULT,S,Y,ALL       ! Item = S,    Comp = Y
OSRESULT,S,EQV,ALL     ! Item = S,    Comp = EQV
OSRESULT,EPPL,INT,ALL  ! Item = EPPL, Comp = INT
...

/POST1
! When issuing postprocessing commands to retrieve selected
!   results (SRES), specify ItemComp (Item + Comp combined)
!   from OSRESULT.
PRESOL,SRES,SY
PRESOL,SRES,SEQV
PRNSOL,SRES,EPPLINT
...
```

Menu Paths

This command cannot be accessed from a menu.

OUTAERO, *SENAME*, *TIMEB*, *DTIME*

Outputs the superelement matrices and load vectors to formatted files for aeroelastic analysis.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SENAME

Name of the superelement that models the wind turbine supporting structure. Defaults to the current Jobname.

TIMEB

First time at which the load vector is formed (defaults to be read from SENAME.sub).

DTIME

Time step size of the load vectors (defaults to be read from SENAME.sub).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Both TIMEB and DTIME must be blank if the time data is to be read from the SENAME.sub file.

The matrix file (SENAME.sub) must be available from the substructure generation run before issuing this command. This superelement that models the wind turbine supporting structure must contain only one master node with six freedoms per node: UX, UY, UZ, ROTX, ROTY, ROTZ. The master node represents the connection point between the turbine and the supporting structure.

This command will generate four files that are exported to the aeroelastic code for integrated wind turbine analysis. The four files are Jobname.GNK for the generalized stiffness matrix, Jobname.GNC for the generalized damping matrix, Jobname.GNM for the generalized mass matrix and Jobname.GNF for the generalized load vectors.

For detailed information on how to perform a wind coupling analysis, see Coupling to External Aeroelastic Analysis of Wind Turbines in the *Advanced Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

OUTGEOM, *Item*, *Freq*

Controls geometry-related data written to the results file.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Geometry data item for file write control:

MAT

Material Properties.

ERASE

Reset **OUTGEOM (p. 1330)** specifications to their default values.

STAT

List the current **OUTGEOM (p. 1330)** specifications.

Freq

Specifies how often to write the specified geometry data:

NONE

Suppress writing of the specified item for all substeps.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Write the data of the specified item for every substep.

Command Default

All geometry data (including material data) is written to the results file.

Notes

The **OUTGEOM (p. 1330)** command controls writing of the specified geometry *Item* to the results file. The geometry items correspond to the geometry records that are included in the results file (see the GEO records of the results file as described in Results File Format in the *Programmer's Reference*).

The command generates a specification for controlling data storage by either activating storage of the specified geometry item (*Freq* = ALL) or by suppressing storage of that item (*Freq* = NONE).

You can issue multiple **OUTGEOM (p. 1330)** commands in an analysis. After the initial command creates the storage specification, subsequent **OUTGEOM (p. 1330)** commands *modify* the specification set. The command processes your specifications in the order in which you input them. If you specify a given *Item* twice, output is based upon the *last* specification.

In addition to **OUTGEOM (p. 1330)**, **OUTPR (p. 1332)** and **OUTRES (p. 1336)** also control solution output. You can issue up to 50 of these output-control commands (any combination of the three) in an analysis.

OUTGEOM (p. 1330),ERASE erases the existing output specifications and resets the counted number of **OUTGEOM (p. 1330)** commands to zero.

Important:

When material property information is not written to the results file (**OUTGEOM (p. 1330)**,MAT,NONE), clearing the database via **/CLEAR (p. 350)** and reading in a set of data in the general postprocessor (**/POST1 (p. 1461)**) via the **SET (p. 1724)** command results in no material property data being stored in the database. In this case, the lack of material data prevents a successful solve from occurring with the modified database, and the results file is only applicable for carrying out post-processing.

The **OUTGEOM (p. 1330)** command is also valid in **/PREP7 (p. 1479)**.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OUTOPT

Specifies "Output options" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Output Options

OUTPR, *Item*, *Freq*, *Cname*

Controls the solution printout.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Item for print control:

BASIC

Basic quantities (nodal DOF solution, nodal reaction loads, and element solution) (default).

NSOL

Nodal DOF solution.

RSOL

Nodal reaction loads.

ESOL

Element solution.

NLOAD

Element nodal loads. When nonlinear stabilization is active, the stabilization force/moments are also printed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SFOR

Stabilization force/moment at the applicable nodes (valid only when nonlinear stabilization is active).

VENG

Element energies. When nonlinear stabilization is active, the energy dissipation due to stabilization is also printed.

RSFO

Result section force/moment output (valid only when a result section is defined). Result section output is always written to a file named `Jobname.secf`.

V

Nodal velocity (applicable to structural transient analysis only (**ANTYPE (p. 162)**,TRANS)).

A

Nodal acceleration (applicable to structural transient analysis only (**ANTYPE (p. 162)**,TRANS)).

ALL

All of the above solution items.

Freq

Print solution for this item every *Freq*th (and the last) substep of each load step. If -*n*, print up to *n* equally spaced solutions (only applies to static or full transient analyses when automatic time stepping is enabled). If NONE, suppress all printout for this item for this load step. If ALL, print solution for this item for every substep. If LAST, print solution for this item only for the last substep of each load step. For a modal analysis, use NONE or ALL.

Cname

Name of the component, created with the **CM (p. 356)** command, defining the selected set of nodes or elements for which this specification is active. If blank, the set is all entities.

Note:

The component named must be of the same type as the item, i.e. nodal or element. A component name is not allowed with the BASIC, RSFO, or ALL labels.

Command Default

No printout.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Controls the solution items to be printed, the frequency with which they are printed (in static, transient, or harmonic analyses), and the set of nodes or elements to which this specification applies (in static, full transient, or full harmonic analyses). An item is associated with either a node (**NSOL (p. 1281)**, **RFORCE (p. 1607)**, V, and A items) or an element (all of the remaining items). The specifications are processed in the order that they are input. Use **OUTPR (p. 1332)**,STAT to list the current specifications and use **OUTPR (p. 1332)**,ERASE to erase all the current specifications.

In addition to **OUTPR (p. 1332)**, **OUTRES (p. 1336)** and **OUTGEOM (p. 1330)** also control solution output. You can issue up to 50 of these output-control commands (any combination of the three) in an analysis.

As described above, **OUTPR (p. 1332)** writes some or all items (depending on analysis type) for all elements. To restrict the solution printout, use **OUTPR (p. 1332)** to selectively suppress (*Freq* = NONE) the writing of solution data, or first suppress the writing of all solution data (**OUTPR (p. 1332)**,ALL,NONE) and then selectively turn on the writing of solution data with subsequent **OUTPR (p. 1332)** commands.

If the generalized plane strain feature is active and **OUTPR (p. 1332)** is issued, the change of fiber length at the ending point during deformation and the rotation of the ending plane about X and Y during deformation will be printed if any displacement at the nodes is printed. The reaction forces at the ending point will be printed if any reaction force at the nodes is printed.

Nodal reaction loads (*Item* = RSOL) are processed according to the specifications listed for the **PRRSOL (p. 1518)** command.

Result printouts for interactive sessions are suppressed for models with more than 10 elements except when the printout is redirected to a file using the **/OUTPUT (p. 1334)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>Solu Printout

Main Menu>Solution>Load Step Opts>Output Ctrls>Solu Printout

/OUTPUT, *Fname*, *Ext*, --, *Loc*

Redirects text output to a file or to the screen.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

Filename and directory path (248 character maximum, including directory) to which text output will be redirected (defaults to Jobname if *Ext* is specified). For interactive runs, *Fname* = TERM (or blank) redirects output to the screen. For batch runs, *Fname* = blank (with all remaining command arguments blank) redirects output to the default system output file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Loc

Location within a file to which output will be written:

(blank)

Output is written starting at the top of the file (default).

APPEND

Output is appended to the existing file.

Command Default

Text output is written to the screen for interactive runs and to the system output file for batch runs (see the *Basic Analysis Guide*).

Notes

Text output includes responses to every command and GUI function, notes, warnings, errors, and other informational messages. Upon execution of **/OUTPUT (p. 1334)**,*Fname*, *Ext*, `...`, all subsequent text output is redirected to the file *Fname*`.`*Ext*. To redirect output back to the default location, issue **/OUTPUT (p. 1334)** (no arguments).

Note:

When using the GUI, output from list operations (**NLIST (p. 1239)**, **DLIST (p. 550)**, etc.) is always sent to a list window regardless of the **/OUTPUT (p. 1334)** setting. The output can then be saved on a file or copied to the **/OUTPUT (p. 1334)** location using the *File* menu in the list window.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Switch Output to>File

Utility Menu>File>Switch Output to>Output Window

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OUTRES,*Item*,*Freq*,*Cname*,--,*NSVAR*,*DSUBres*

Controls the solution-result data written to the database.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Solution item for database and file-write control:

ALL

All solution items except LOCI, SVAR, and NAR (default).

BASIC

Write only NSOL, RSOL, NLOAD, STRS, FGRAD, and FFLUX records to the results file and database.

ERASE

Resets **OUTRES (p. 1336)** specifications to their default values.

STAT

Lists the current **OUTRES (p. 1336)** specifications.

NSOL

Nodal degree-of-freedom solution.

RSOL

Nodal reaction loads.

V

Nodal velocity (applicable to structural transient analysis only (**ANTYPE (p. 162)**,TRANS)).

A

Nodal acceleration (applicable to structural transient analysis only (**ANTYPE (p. 162)**,TRANS)).

CINT

All available results generated by **CINT (p. 334)**.

SVAR

State variables (used with supported subroutines that customize material behavior).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ESOL

Enables or disables *all* of the following element-solution items. (These items can also be individually enabled or disabled.)

NLOAD

Element nodal, input constraint, and force loads (also used with POST1 commands **PRRFOR (p. 1517)**, **NFORCE (p. 1216)**, and **FSUM (p. 780)** to calculate reaction loads).

STRS

Element nodal stresses.

EPEL

Element elastic strains.

EPTH

Element thermal, initial, and swelling strains.

EPPL

Element plastic strains.

EPCR

Element creep strains.

EPDI

Element diffusion strains.

FGRAD

Element nodal gradients.

FFLUX

Element nodal fluxes.

LOCI

Integration point locations.

VENG

Element energies.

MISC

Element miscellaneous data (**ETABLE (p. 687)** SMISC and NMISC items).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FMAG

Electromagnetic nodal forces.

CURD

Element source current density.

NLDAT

Element nonlinear data.

EHEAT

Element heat generation rate.

ETMP

Element temperatures.

SRFS

Element surface stresses.

CONT

Element contact data.

BKSTR

Element backstresses. Enabling this output also requires that you enable EPPL.

EANGL

Element Euler angles.

AESO

Enables or disables *all* of the following advanced element-solution output items. (These items *cannot* be individually enabled or disabled.)

BKS - Back-stress for kinematic hardening.

CDM - Damage variable for Mullins effect.

ESIG - BIOT's effective stress.

FFLX - Fluid flow flux in poromechanics.

FICT - Fictive temperature.

FSVAR - Fluence state variables.

MPLS - Microplane damage.

NS - Nominal strain.

PDMG - Progressive damage parameters.

PFC - Failure criteria based on the effective stresses in the damaged material.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PMSV - Permeability state variables.

SEND - Energy record.

TF - Thermal flux.

TG - Thermal gradient.

YSIDX - Yield status for geomechanical materials.

NAR

Enables or disables *all* of the following nodal-averaged solution items. (These items can also be individually enabled or disabled.)

NDST

Nodal-averaged stresses.

NDEL

Nodal-averaged elastic strains.

NDPL

Nodal-averaged plastic strains.

NDCR

Nodal-averaged creep strains.

NDTH

Nodal-averaged thermal and swelling strains.

DSUB

Enables `Jobname.dsub` file writing for all superelements if the nodal degree-of-freedom solution (*Item* = NSOL) file writing is disabled (see notes for details and an example).

Freq

Specifies how often (that is, at which substeps) to write the specified solution item.

LAST

Writes the specified solution item only for the last substep of each load step (default).

n

Writes the specified solution item every *n*th (and the last) substep of each load step.

-n

Writes up to *n* equally spaced solutions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The time range is divided into equally spaced time points. The first solution at or just past the time point is written. If the time steps span two time points, only one is written.

NONE

Suppresses writing of the specified solution item for all substeps.

ALL

Writes the solution of the specified solution item for every substep. This value is the default for a harmonic analysis (**ANTYPE (p. 162)**,HARMIC) and for any expansion pass (**EXPASS (p. 714)**,ON).

%*array*%

Where *array* is the name of an *n* x 1 x 1 dimensional array parameter defining *n* key times, the data for the specified solution item is written at those key times.

Key times in the array parameter must appear in ascending order. Values must be greater than or equal to the beginning values of the load step, and less than or equal to the ending time values of the load step.

For multiple-load-step problems, either change the parameter values to fall between the beginning and ending time values of the load step or erase the current settings and reissue the command with a new array parameter.

For more information about defining array parameters, see ***DIM (p. 530)**.

Cname

The name of the component (created via **CM (p. 356)**) defining the selected set of elements or nodes for which this specification is active. If not specified, the set is *all* entities. A component name is not allowed with the ALL, BASIC, or RSOL items.

--

Reserved.

NSVAR

The number of user-defined state variables (**TB (p. 1899)**,STATE) to be written to the results file. Valid only when *Item* = SVAR and user-defined state variables exist. The specified value cannot exceed the total number of state variables defined; if no value is specified, *all* user-defined state variables are written to the results file. This argument acts on all sets of user-defined state variables that exist for the model.

DSUBres

Specifies whether to write additional results in `Jobname.dsub` during a substructure or CMS use pass in a transient or harmonic analysis.

Blank

Write the nodal degree-of-freedom solution in `Jobname.dsub` (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

In addition to the nodal degree-of-freedom solution, also write necessary data to compute quantities using nodal velocity and nodal acceleration (damping force, inertial force, kinetic energy, etc.) in the subsequent expansion pass. For more information, see Step 3: Expansion Pass in the *Substructuring Analysis Guide*.

Command Default

Writes the specified solution item for every substep. Exceptions to the default behavior are as follows:

- For static (**ANTYPE (p. 162)**,STATIC) or transient (**ANTYPE (p. 162)**,TRANS) analyses, the default is to write the specified solution item for the last substep of each load step.
- For a harmonic analysis (**ANTYPE (p. 162)**,HARMIC) and any expansion pass, the default is to write the specified solution item for every substep.
- For a mode-superposition transient analysis, the default is to write the reduced displacements file for every fourth substep (as well as the last substep).

Notes

OUTRES (p. 1336) controls following output parameters:

- The solution item (*Item*) to write to the database (and to the reduced displacement and results files)
- The frequency (*Freq*) at which the solution item is written (applicable to static, transient, or full harmonic analyses)
- The set of elements or nodes (*Cname*) to which your specification applies.

The command generates a specification for controlling data storage for each substep, activating storage of the specified solution item for the specified substeps of the solution *and* suppressing storage of that item for *all other substeps*.

You can issue **OUTRES (p. 1336)** multiple times in an analysis. After the initial command creating the storage specification, subsequent **OUTRES (p. 1336)** commands *modify* the specification set for each substep. The command processes your specifications at each substep in the order in which you input them. If you specify a given solution item twice, output is based upon the *last* specification. Therefore, issue multiple **OUTRES (p. 1336)** commands carefully and in the proper sequence.

In addition to **OUTRES (p. 1336)**, these commands also control solution output: **OUTPR (p. 1332)**, **OUTGEOM (p. 1330)**, and **OSRESULT (p. 1327)**.

You can issue up to 50 output-control commands for **OUTRES (p. 1336)**, **OUTPR (p. 1332)**, **OUTGEOM (p. 1330)** in an analysis. There is no limit on the number of **OSRESULT (p. 1327)** commands.

OUTRES (p. 1336),ERASE erases the existing output specifications and resets the counted number of **OUTRES (p. 1336)** commands to zero. The ERASE argument works in a similar manner for **OUTPR (p. 1332)**, **OUTGEOM (p. 1330)**, and **OSRESULT (p. 1327)**.

A given **OUTRES (p. 1336)** command generally has no effect on solution items *not* specified. For example, an **OUTRES (p. 1336)**,ESOL,LAST command does not affect NSOL data; that is, it neither activates nor

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

suppresses NSOL data storage in any substep. An exception to this behavior involves the EANGL solution item.

OUTRES (p. 1336) controls element Euler angle (EANGL) data output as follows:

- When *Freq* = NONE, no element Euler angles are output at any substeps.

  Without Euler angles, element results postprocessing can occur in the element solution coordinate system only (that is, **RSYS (p. 1639)** has no effect on element results); therefore, nodal averaging of element solution items may not be applicable when element solution coordinate systems are not uniform.

- When *Freq* = any other value (including the command default), element Euler angles are output at substeps specified by *Freq*, *and* at any substeps where one or more tensorial element solution items (STRS, EPEL, EPTH, EPPL, EPCR, EPDI, FGRAD, FFLUX, and AESO) are output.

Additional results in the Jobname.dsub file (*DSUBres* = ALL) can only be requested in the first load step.

Important:

In the results-item hierarchy, certain items are subsets of other items. For example, element solution (ESOL) data is a subset of all (ALL) solution data. An **OUTRES (p. 1336)**,ALL command can therefore affect ESOL data. Likewise, an **OUTRES (p. 1336)** command that controls ESOL data can affect a *portion* of all data.

The example **OUTRES (p. 1336)** commands (p. 1343) illustrate the interrelationships between solution items and why it is necessary to issue **OUTRES (p. 1336)** thoughtfully.

To suppress all data at every substep, issue the **OUTRES (p. 1336)**,ALL,NONE command. (**OUTRES (p. 1336)**,ERASE *does not* suppress all data at every substep.)

The NSOL, RSOL, V, and A solution items are associated with nodes. The CINT solution item is associated with fracture. All remaining solution items are associated with elements.

Enabling nodal-averaged results (*Item* = NAR or any of the associated labels) generally reduces the results file size, provided the equivalent element-based results are concurrently disabled. When nodal-averaged results are enabled, element values for stress and strain are averaged and stored as nodal values. Some limitations apply when using nodal averaged results. For more information and an example, see Nodal-Averaged Results in the *Element Reference*.

The boundary conditions (constraints and force loads) are written to the results file only if either nodal or reaction loads (NLOAD or RSOL items) are also written.

Usage considerations when specifying *Freq*:

- The only valid labels for *Freq* are NONE or ALL for the following:
  - Modal analysis
  - Component mode synthesis (CMS) generation pass with *Elcalc* = YES on the **CMSOPT (p. 378)** command

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Any expansion pass (**EXPASS (p. 714)**,ON)

- When superelements exist in the model, the *Freq* specified for *Item* = NSOL controls the write frequency of the nodal degree-of-freedom solution to the Jobname.rst file and the superelements reduced nodal degree-of-freedom solution to the Jobname.dsub file. The program only considers a *Freq* specification for *Item* = DSUB if file writing for *Item* = NSOL is disabled.

  Example 11: To enable Jobname.dsub file writing (*Item* = DSUB at a specified *Freq*), first disable nodal DOF solution writing to the Jobname.rst file (*Item* = NSOL)

```
OUTRES,ERASE        !reset specifications to default values
OUTRES,ALL,NONE     !disable writing of all solution items, including NSOL
OUTRES,DSUB,ALL     !enable Jobname.dsub file writing at every substep
```

  After issuing the above commands, the nodal degree-of-freedom solutions are not written to Jobname.rst, but the reduced nodal degree-of-freedom solutions are written to Jobname.dsub for every substep.

For additive manufacturing analyses during the build step (**AMSTEP (p. 124)**,BUILD), *Freq* refers to the layer number (for example, output ALL layers, LAST layer, or every N^{th} layer).

To specify selected results to output to the database, see **OSRESULT (p. 1327)**.

OUTRES (p. 1336) is also valid in PREP7.

Example

When issuing an **OUTRES (p. 1336)** command, think of a matrix in which you set switches on and off. When a switch is on, a solution item is stored for the specified substep. When a switch is off, a solution item is suppressed for a specified substep.

Assuming a static (**ANTYPE (p. 162)**,STATIC) analysis, this example shows how the matrix looks after issuing each **OUTRES (p. 1336)** command in this six-substep solution.

```
NSUBST,6
OUTRES,ERASE
OUTRES,NSOL,2
OUTRES,ALL,3
OUTRES,ESOL,4
SOLVE
```

To simplify the example, only a subset of the available solution items appears in the matrix.

OUTRES (p. 1336),ERASE -- After issuing this command, the default output specifications are in effect, as shown:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	off	off	off	off	off	off	off
3	off	off	off	off	off	off	off
4	off	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**

OUTRES (p. 1336),NSOL,2 -- This command modifies the initial specifications so that NSOL is enabled for substeps 2, 4 and 6, and disabled for substeps 1, 3 and 5, as shown:

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	**ON**	off	off	off	off	off	off
3	off	off	off	off	off	off	off
4	**ON**	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

OUTRES (p. 1336),ALL,3 -- This command further modifies the specifications so that ALL is enabled for substeps 3 and 6, and disabled for substeps 1, 2, 4 and 5, as shown:

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	**off**	off	off	off	off	off	off
3	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**	**ON**
4	**off**	off	off	off	off	off	off
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OUTRES (p. 1336),ESOL,4 -- This command once again modifies the specifications so that ESOL is enabled for the fourth and last substeps, and disabled for substeps 1, 2, 3 and 5, as shown:

Substep	Results Item Specification						
	ALL						
	BASIC						
	NSOL	RSOL	ESOL				
			NLOAD	STRS	FGRAD	EPEL	EPTH
1	off	off	off	off	off	off	off
2	off	off	off	off	off	off	off
3	ON	ON	**off**	**off**	**off**	**off**	**off**
4	off	off	**ON**	**ON**	**ON**	**ON**	**ON**
5	off	off	off	off	off	off	off
6	ON	ON	ON	ON	ON	ON	ON

SOLVE (p. 1822)

When obtaining the solution, results data are stored as follows:

Substep	Results Items Stored
1	*No data*
2	*No data*
3	NSOL and RSOL data
4	ESOL data
5	*No data*
6	ALL data

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Preprocessor>Loads>Load Step Opts>Output Ctrls>DB/Results File

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Load Step Opts>Output Ctrls>DB/Results File

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

P Commands

PADELE, *DELOPT*

Deletes a defined path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DELOPT

Path delete option (one of the following):

ALL

Delete all defined paths.

NAME

Delete a specific path from the list of path definitions. (Substitute the actual path name for NAME.)

Command Default

Deletes the currently active path.

Notes

Paths are identified by individual path names. To review the current list of path names, issue the command **PATH (p. 1354)**,STATUS.

This command is valid in the general postprocessor.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array

Main Menu>General Postproc>Path Operations>Delete Path>All Paths

Main Menu>General Postproc>Path Operations>Delete Path>By Name

Main Menu>Preprocessor>Path Operations>Delete Path>All Paths

Main Menu>Preprocessor>Path Operations>Delete Path>By Name

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/PAGE, *ILINE*, *ICHAR*, *BLINE*, *BCHAR*, *COMMA*

Defines the printout and screen page size.

POST1 (p. 51): Listing (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ILINE

Number of lines (11 minimum) per "page" or screen. Defaults to 24. Applies to interactive non-GUI to the screen output only.

ICHAR

Number of characters (41 to 132) per line before wraparound. Defaults to 80. Applies to interactive non-GUI to the screen output only.

BLINE

Number of lines (11 minimum) per page. Defaults to 56. Applies to batch mode (**/BATCH (p. 217)**), diverted (**/OUTPUT (p. 1334)**), or interactive GUI (**/MENU (p. 1121)**) output. If negative, no page headers are output.

BCHAR

Number of characters (41 to 240 (system dependent)) per line before wraparound. Defaults to 140. Applies to batch mode (**/BATCH (p. 217)**), diverted (**/OUTPUT (p. 1334)**), or interactive GUI (**/MENU (p. 1121)**) output.

COMMA

Input 1 to specify comma-separated output for node (**NLIST (p. 1239)**) and element (**ELIST (p. 619)**) output.

Command Default

As defined by the items above.

Notes

Defines the printout page size for batch runs and the screen page size for interactive runs. Applies to the POST1 **PRNSOL (p. 1505)**, **PRESOL (p. 1481)**, **PRETAB (p. 1490)**, **PRRSOL (p. 1518)**, and **PRPATH (p. 1517)** commands. See the **/HEADER (p. 902)** command for additional controls (page ejects, headers, etc.) that affect the amount of printout. A blank (or out-of-range) value retains the previous setting. Issue **/PAGE (p. 1348)**,STAT to display the current settings. Issue **/PAGE (p. 1348)**,DEFA to reset the default specifications.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

PAGET, *PARRAY*, *P0PT*

Writes current path information into an array variable.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

PARRAY

The name of the array parameter that Mechanical APDL creates to store the path information. If the array parameter already exists, it will be replaced with the current path information.

P0PT

Determines how data will be stored in the parameter specified with *PARRAY*:

POINTS

Store the path points, the nodes (if any), and coordinate system. (For information on defining paths and path points, see the descriptions of the **PATH (p. 1354)** and **PPATH (p. 1463)** commands.)

TABLE

Store the path data items. (See the **PDEF (p. 1372)** command description for path data items.)

LABEL

Stores path data labels.

Notes

Use the **PAGET (p. 1349)** command with the **PAPUT (p. 1350)** command to store and retrieve path data in array variables for archiving purposes.

When retrieving path information, restore the path points (POINTS option) first, then the path data (TABLE option), and then the path labels (LABEL option).

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Store>Path in array

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PAPUT, *PARRAY*, *POPT*

Retrieves path information from an array variable.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

PARRAY

Name of the array variable containing the path information.

POPT

Specifies which path data to retrieve:

POINTS

Retrieve path point information (specified with the **PPATH (p. 1463)** command and stored with the **PAGET (p. 1349)**,POINTS command). The path data name will be assigned to the path points.

TABLE

Retrieve path data items (defined via the **PDEF (p. 1372)** command and stored with the **PAGET (p. 1349)**,,TABLE command).

LABEL

Retrieve path labels stored with the **PAGET (p. 1349)**,,LABEL command.

Notes

When retrieving path information, restore path points (POINTS option) first, then the path data (TABLE option), and then the path labels (LABEL option).

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array

PARESU, *Lab*, *Fname*, *Ext*, - -

Restores previously saved paths from a file.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Read operation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Read all paths from the selected file (default).

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to PATH if *Fname* is blank.

--

Unused field.

Notes

This command removes all paths from virtual memory and then reads path data from a file written with the **PASAVE (p. 1353)** command. All paths on the file will be restored. All paths currently in memory will be deleted.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Paths from file

Main Menu>Preprocessor>Path Operations>Retrieve>Paths from file

---

PARRES, *Lab*, *Fname*, *Ext*, *--*

Reads parameters from a file.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Lab

Read operation:

NEW --

Replace current parameter set with these parameters (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CHANGE --

Extend current parameter set with these parameters, replacing any that already exist.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

The extension defaults to PARM if *Fname* is blank.

--

Unused field.

Notes

Reads parameters from a coded file. The parameter file may have been written with the **PARSAV (p. 1352)** command. The parameters read may replace or change the current parameter set.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Restore Parameters

PARSAV, *Lab*, *Fname*, *Ext*, *--*

Writes parameters to a file.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Lab

Write operation:

SCALAR --

Write only scalar parameters (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL --

Write scalar and array parameters. Parameters may be numeric or alphanumeric.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

The extension defaults to PARM if *Fname* is blank.

--

Unused field.

Notes

Writes the current parameters to a coded file. Previous parameters on this file, if any, will be overwritten. The parameter file may be read with the **PARRES (p. 1351)** command.

PARSAV (p. 1352)/**PARRES (p. 1351)** operations truncate some long decimal strings, and can cause differing values in your solution data when other operations are performed. A good practice is to limit the number of decimal places you will use before and after these operations.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Save Parameters

PASAVE, *Lab*, *Fname*, *Ext*, *--*

Saves selected paths to an external file.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Write operation:

S

Saves only selected paths.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Saves all paths (default).

Pname

Saves the named path (from the **PSEL (p. 1536)** command).

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to `Jobname`.

Ext

Filename extension (eight-character maximum).

The extension defaults to PATH if *Fname* is blank.

--

Unused field.

Notes

Saves the paths selected with the **PSEL (p. 1536)** command to an external file (default `Jobname.path`).

Previous paths on this file, if any, are overwritten. The path file can be read via **PARESU (p. 1350)**.

This command is valid in POST1.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Store>Paths in file

Main Menu>Preprocessor>Path Operations>Store>Paths in file

PATH,*NAME*,*nPts*,*nSets*,*nDiv*

Defines a path name and establishes parameters for the path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NAME

Name for this path (eight characters maximum. If *nPts* is blank, set the current path to the path with this name. If *nPts* is greater than zero, create a path of this name. If a path with this name already exists, replace it with a new path. If the *NAME* value is STATUS, display the status for path settings.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nPts

The number of points used to define this path. The minimum number is two, and the maximum is 1000. Default is 2.

nSets

The number of sets of data which you can map to this path. You must specify at least four: X, Y, Z, and S. Default is 30.

nDiv

The number of divisions between adjacent points. Default is 20. There is no maximum number of divisions.

Notes

The **PATH (p. 1354)** command is used to define parameters for establishing a path. The path geometry is created by the **PPATH (p. 1463)** command. Multiple paths may be defined and named; however, only one path may be active for data interpolation (**PDEF (p. 1372)**) and data operations (**PCALC (p. 1362)**, etc.). Path geometry points and data are stored in memory while in POST1. If you leave POST1, the path information is erased. Path geometry and data may be saved in a file by archiving the data using the **PASAVE (p. 1353)** command. Path information may be restored by retrieving the data using the **PARESU (p. 1350)** command.

For overlapping nodes, the lowest numbered node is assigned to the path.

The number of divisions defined using *nDiv* does NOT affect the number of divisions used by **PLSECT (p. 1440)** and **PRSECT (p. 1520)**.

For information on displaying paths you have defined, see Mapping Results onto a Path in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>List Results>Path Items

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Path from array

Main Menu>General Postproc>Path Operations>Archive Path>Retrieve>Paths from file

Main Menu>General Postproc>Path Operations>Define Path>By Location

Main Menu>General Postproc>Path Operations>Define Path>By Nodes

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>General Postproc>Path Operations>Define Path>Path Status>Defined Paths

Main Menu>General Postproc>Path Operations>Delete Path>All Paths

Main Menu>General Postproc>Path Operations>Delete Path>By Name

Main Menu>General Postproc>Path Operations>Recall Path

Main Menu>Preprocessor>Path Operations>Define Path>By Location

Main Menu>Preprocessor>Path Operations>Define Path>By Nodes

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Path Operations>Define Path>Path Status>Defined Paths

Main Menu>Preprocessor>Path Operations>Delete Path>All Paths

Main Menu>Preprocessor>Path Operations>Delete Path>By Name

Main Menu>Preprocessor>Path Operations>Recall Path

Main Menu>Preprocessor>Path Operations>Retrieve>Paths from file

Utility Menu>List>Status>General Postproc>Path Operations

PAUSE

Temporarily releases the current product license.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

The **PAUSE (p. 1356)** command temporarily releases (or pauses) the current product license so that another application can use it.

This application consumes a license as soon as you launch it, and retains that license until it is finished. If you launch the product interactively, the license is retained until you either close the application or issue a **PAUSE (p. 1356)** command via the command line.

No other operation (other than **SAVE (p. 1645)** or **/EXIT (p. 708)**) is possible in the current application while use of the product license is paused.

When the second application has finished and releases the license, issue an **UNPAUSE (p. 2056)** command via the command line to restore use of the license to the current application.

For more information, see the Ansys Licensing Guide.

Menu Paths

This command cannot be accessed from a menu.

/PBC, *Item*, --, *KEY*, *MIN*, *MAX*, *ABS*

Shows boundary condition (BC) symbols and values on displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

U

Applied translational constraints (UX, UY, UZ).

ROT

Applied rotational constraints (ROTX, ROTY, ROTZ).

TEMP

Applied temperatures (TEMP, TBOT, TE2, TE3, . . ., TTOP).

PRES

Applied fluid pressures.

V

Applied flow velocities (VX, VY, VZ).

VOLT

Applied voltages.

MAG

Applied scalar magnetic potentials.

A

Applied vector magnetic potentials.

CONC

Concentration.

CHRG

Applied electric charge.

F or FORC

Applied structural forces (FX, FY, FZ).

M or MOME

Applied structural moments (MX, MY, MZ).

HEAT

Applied heat flows (HEAT, HBOT, HE2, HE3, . . ., HTOP).

FLOW

Applied fluid flow.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AMPS

Applied current flow.

FLUX

Applied magnetic flux.

CSG

Applied magnetic current segments.

RATE

Diffusion flow rate.

MAST

Master degrees of freedom.

CP

Coupled nodes.

CE

Nodes in constraint equations.

NFOR

POST1 nodal forces.

NMOM

POST1 nodal moments

RFOR

POST1 reaction forces.

RMOM

POST1 reaction moments (MX, MY, MZ).

PATH

Path geometry (undistorted) associated with the **PATH (p. 1354)** command after a **PDEF (p. 1372)** or **PVECT (p. 1556)** command has been issued.

ACEL

Global acceleration (ACELX, ACELY, ACELZ vector).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OMEG

Global angular velocity (OMEGX, OMEGY, OMEGZ vector) and acceleration (DOMEGX, DOMEGY, DOMEGZ vector).

ALL

Represents all appropriate labels.

- -

Unused field.

KEY

Symbol key:

0

Do not show symbol.

1

Show symbol.

2

Plot value next to symbol.

MIN

Minimum value in a range of values plotted on screen.

MAX

Maximum value in a range of values plotted on screen.

ABS

Absolute number. If *KEY* = 2 and *ABS* = 0, a number falling between the *MIN* and *MAX* is displayed. If *ABS* is not specified, it defaults to 0. If *KEY* = 2 and *ABS* = 1, an absolute value falling between the *MIN* and *MAX* is displayed. *ABS* = 1 lets you eliminate the display of numbers whose absolute values are less than a desired tolerance. For example, if *ABS* = 1, *MIN* = 10 and *MAX* = 1e8, values such as .83646 and -5.59737 are not displayed.

Command Default

No symbols displayed.

Notes

The **/PBC** command adds degree of freedom constraint, force load, and other symbols to displays.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Symbols are applied to the selected nodes only. All arrow and arrowhead symbols are oriented in the nodal coordinate system and lie in two perpendicular planes. Force arrows are scaled proportional to their magnitude. (If *KEY* = 1, use **/VSCALE (p. 2160)** to change arrow length.) For scalar quantities, the specific component direction (that is, x, y, or z) of the symbol has no meaning, but the positive or negative sense (for example, positive or negative x) represents a positive or negative scalar value, respectively.

The effects of the **/PBC** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PBC** command). If you issue multiple **/PBC** commands during an analysis, only the setting specified by the most recent **/PBC (p. 1356)** command applies.

Use **/PSTATUS (p. 1550)** or **/PBC (p. 1356)**,STAT to display settings. Use **/PBC (p. 1356)**,DEFA to reset all specifications back to default. See the **/PSF (p. 1537)** and **/PBF (p. 1360)** commands for other display symbols.

In a cyclic symmetry analysis, the **/PBC (p. 1356)** command is deactivated when cyclic expansion is active (**/CYCEXPAND (p. 454)**,,ON). To view boundary conditions on the base sector, deactivate cyclic expansion (**/CYCEXPAND (p. 454)**,,OFF) and issue this command: **/PBC (p. 1356)**,ALL,,1

Issuing the command **/PBC (p. 1356)**,PATH, ,1 displays all defined paths.

The **/PBC (p. 1356)** command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>General Postproc>Path Operations>Plot Paths

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Plot Paths

Utility Menu>PlotCtrls>Symbols

/PBF,*Item*,--,*KEY*

Shows magnitude of body-force loads on displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item:

TEMP

Applied temperatures.

FLUE

Applied fluences.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HGEN

Applied heat generation rates.

JS

Applied current density magnitude.

JSX

X-component of current density.

JSY

Y-component of current density.

JSZ

Z-component of current density.

PHASE

Phase angle of applied load.

MVDI

Applied magnetic virtual displacements flag.

CHRGD

Applied electric charge density.

- -

Unused field.

KEY

Symbol key:

0

Do not show body-force load contours.

1

Show body-force load contours.

2

Show current density as a vector (not a contour).

Command Default

No body-force load contours displayed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Shows body-force loads as contours on displays for the selected elements.

The effects of the **/PBF** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PBF** command). If you issue multiple **/PBF** commands during an analysis, only the setting specified by the most recent **/PBF (p. 1360)** command applies.

Use **/PSTATUS (p. 1550)** or **/PBF (p. 1360)**,STAT to display settings. Use **/PBF (p. 1360)**,DEFA to reset all specifications back to default. See also the **/PSF (p. 1537)** and **/PBC (p. 1356)** command for other display contours.

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Symbols

PCALC, *Oper*, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*, *CONST*

Forms additional labeled path items by operating on existing path items.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Type of operation to be performed. See "Notes" (p. 1363) below for specific descriptions of each operation:

ADD

Adds two existing path items.

MULT

Multiplies two existing path items.

DIV

Divides two existing path items (a divide by zero results in a value of zero).

EXP

Exponentiates and adds existing path items.

DERI

Finds a derivative.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

INTG

Finds an integral.

SIN

Sine.

COS

Cosine.

ASIN

Arcsine.

ACOS

Arccosine.

LOG

Natural log.

LabR

Label assigned to the resulting path item.

Lab1

First labeled path item in operation.

Lab2

Second labeled path item in operation. *Lab2* must not be blank for the MULT, DIV, DERI, and INTG operations.

FACT1

Factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2

Factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

CONST

Constant value (defaults to 0.0).

Notes

If *Oper* = ADD, the command format is:

PCALC (p. 1362),ADD,*LabR*,*Lab1*,*Lab2*,*FACT1*,*FACT2*,*CONST*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This operation adds two existing path items according to the operation:

LabR = (*FACT1* x *Lab1*) + (*FACT2* x *Lab2*) + *CONST*

It may be used to scale the results for a single path item.

If *Oper* = MULT, the command format is:

PCALC (p. 1362),MULT,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation multiplies two existing path items according to the operation:

LabR = *Lab1* x *Lab2* x *FACT1*

If *Oper* = DIV, the command format is:

PCALC (p. 1362),DIV,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation divides two existing path items according to the operation:

LabR = (*Lab1*/*Lab2*) x *FACT1*

If *Oper* = EXP, the command format is:

PCALC (p. 1362),EXP,*LabR*,*Lab1*,*Lab2*,*FACT1*,*FACT2*

This operation exponentiates and adds existing path items according to the operation:

LabR = ($|Lab1|^{FACT1}$) + ($|Lab2|^{FACT2}|$)

If *Oper* = DERI, the command format is:

PCALC (p. 1362),DERI,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation finds a derivative according to the operation:

LabR = *FACT1* x d(*Lab1*)/d(*Lab2*)

If *Oper* = INTG, the command format is:

PCALC (p. 1362),INTG,*LabR*,*Lab1*,*Lab2*,*FACT1*

Lab2 must not be blank. This operation finds an integral according to the operation:

$$LabR = FACT1 \times \int_S Lab1\, d(Lab2)$$

Use S for *Lab2* to integrate *Lab1* with respect to the path length. S, the distance along the path, is automatically calculated by the program when a path item is created with the **PDEF (p. 1372)** command.

If *Oper* = SIN, COS, ASIN, ACOS, or LOG, the command format is:

PCALC (p. 1362),Oper,*LabR*,*Lab1*,,*FACT1*,*CONST*

where the function (SIN, COS, ASIN, ACOS or LOG) is substituted for *Oper* and *Lab2* is blank.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The operation finds the resulting path item according to one of the following formulas:

$LabR$ = FACT2 x $sin(FACT1 \times Lab1) + CONST$

$LabR$ = FACT2 x $cos(FACT1 \times Lab1) + CONST$

$LabR$ = FACT2 x $sin^{-1}(FACT1 \times Lab1) + CONST$

$LabR$ = FACT2 x $cos^{-1}(FACT1 \times Lab1) + CONST$

$LabR$ = FACT2 x $log(FACT1 \times Lab1) + CONST$

Menu Paths

Main Menu>General Postproc>Path Operations>Add
Main Menu>General Postproc>Path Operations>ArcCosine
Main Menu>General Postproc>Path Operations>ArcSine
Main Menu>General Postproc>Path Operations>Cosine
Main Menu>General Postproc>Path Operations>Differentiate
Main Menu>General Postproc>Path Operations>Divide
Main Menu>General Postproc>Path Operations>Exponentiate
Main Menu>General Postproc>Path Operations>Integrate
Main Menu>General Postproc>Path Operations>Multiply
Main Menu>General Postproc>Path Operations>Natural Log
Main Menu>General Postproc>Path Operations>Sine

PCGOPT,*Lev_Diff*,*Fallback*,*ReduceIO*,*StrmCk*,*Wrtfull*,*LM_Key*

Controls PCG solver options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lev_Diff

Indicates the level of difficulty of the analysis. Valid settings are AUTO or 0 (default), 1, 2, 3, 4, or 5. This option applies to both the PCG solver when used in a static, full transient, or full harmonic analyses and to the PCG Lanczos method in modal analyses.

Specify AUTO to allow Mechanical APDL to select the proper level of difficulty for the model.

Lower values (1 or 2) generally provide the best performance for well-conditioned problems.

Values of 3 or 4 generally provide the best performance for ill-conditioned problems; however, higher values may increase the solution time for well-conditioned problems. Higher level-of-difficulty values typically require more memory.

The highest value of 5 essentially performs a factorization of the global matrix (similar to the sparse solver) and may require a very large amount of memory. This level is generally recom-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mended for small- to medium-sized problems when using the PCG Lanczos mode-extraction method. *Lev_Diff* = 5 is not supported for a full harmonic analysis.

For example, models containing elongated elements (that is, elements with high aspect ratios) and models containing contact elements can lead to ill-conditioned problems. To determine if your problem is ill-conditioned, view the Jobname.pcs file to see the number of PCG iterations needed to reach a converged solution. Generally, static, full transient, or full harmonic solutions that require more than 1500 iterations are considered to be ill-conditioned for the PCG solver.

Fallback

Controls whether Mechanical APDL switches to the sparse direct solver automatically (**EQSLV (p. 648)**,SPARSE) under certain conditions. (The criteria are listed below.) When Mechanical APDL switched the equation solver, the simulation attempts to continue without interruption.

AUTO

Automatically switch to the sparse solver when one of the following conditions apply (default):

- The assembled matrix is detected to be indefinite.
- The PCG solver requires more than 2000 iterations to reach convergence.
- The PCG solver fails to converge.

ON

More aggressive fallback criteria. Automatically switch to the sparse solver when one of the following conditions apply:

- The assembled matrix is detected to be indefinite.
- The PCG solver requires more than 1500 iterations to reach convergence.
- The PCG solver fails to converge.

OFF

Disables the fallback logic so that there is no automatic switching, and only the PCG solver is used to solve the equations for this simulation.

After switching to the sparse solver, the program reverts back to the PCG solver under certain conditions. See the Notes (p. ?) section for details and limitations.

ReduceIO

Controls whether the PCG solver will attempt to reduce I/O performed during equation solution:

AUTO

Automatically chooses whether to reduce I/O or not (default).

YES

Reduces I/O performed during equation solution in order to reduce total solver time.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NO

Does NOT reduce I/O performed during equation solution.

This option applies to both the PCG solver when used in static and full transient analyses and to the PCG Lanczos method in modal analyses.

StrmCk

Controls whether or not a Sturm sequence check is performed:

OFF

Does NOT perform Sturm sequence check (default).

ON

Performs Sturm sequence check

This option applies only when using the PCG Lanczos method in modal analyses. When using this option, a factorization must be performed and will require a very large amount of memory for extra computations. This option is generally recommended for small- to medium-sized problems. If the Sturm sequence check takes a large amount of computing time, use the `Jobname.ABT` file to abort the Sturm check, or press the **STOP** button if in interactive mode.

Wrtfull

Controls whether or not the `.full` file is written.

ON

Write `.full` file (default)

OFF

Do not write `.full` file.

This option applies only when using the PCG Lanczos method in modal analyses because the `.full` file is never written when using the PCG solver in static, full transient, or harmonic analyses.

If using **MSAVE (p. 1183)**,ON and conditions for the **MSAVE (p. 1183)** command are met, a complete `.full` file is never written regardless of this option.

If constraint equations are present in the model, a `.full` file is always written regardless of this option.

This option is useful in a distributed-memory parallel processing analysis because assembling the global stiffness and mass matrices on the head compute node before writing the `.full` file can take a considerable amount of memory. By setting *Wrtfull* = OFF, this assembly process is skipped on the head compute node, decreasing the amount of memory required to compute the modes and mode shapes. *Wrtfull* = OFF does not affect the results for the modes and mode shapes. However, without a `.full` file, the participation factor table computations do not occur.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To generate the `.full` file to be used in a downstream harmonic, transient mode-superposition, or spectrum analysis, rerun the modal analysis with *`Wrtfull`* = ON, or use the **WRFULL (p. 2184)** command.

`LM_Key`

Controls use of the PCG solver for MPC184 elements that involve the Lagrange multiplier method. This option applies only to the PCG solver when used in static analyses, full transient analyses, and modal analyses that use the PCG Lanczos mode-extraction method (**MODOPT (p. 1140)**,LANPCG).

ON

Allow use of the PCG solver with certain MPC184 element types that use the Lagrange multiplier method. (default)

OFF

Do not use the PCG solver with any MPC184 element types that use the Lagrange multiplier method.

The Lagrange multiplier method used by MPC184 elements transfers the Lagrange multipliers into multiple point constraints and, hence, can be solved by the PCG solver. The current MPC184 element types supported are: rigid beam, rigid link, slider, revolute joint, universal joint, translational joint, cylindrical joint, weld joint, spherical joint, and general joint. For all other MPC184 element types, the PCG solver cannot be used, and the equation solver automatically switches to the sparse solver regardless of the *`LM_Key`* setting on **PCGOPT (p. 1365)**.The **MSAVE (p. 1183)** command does not support the *`LM_Key`* = ON option.

Notes

`ReduceIO` works independently of the **MSAVE (p. 1183)** command in the PCG solver. Setting *`ReduceIO`* to YES can significantly increase the memory usage in the PCG solver.

To minimize the memory used by the PCG solver with respect to the *`Lev_Diff`* option only, set *`Lev_Diff`* = 1 if you do not have sufficient memory to run the PCG solver with *`Lev_Diff`* = AUTO.

For unsymmetric structural and harmonic analyses, the PCG solver supports a *`Lev_Diff`* equal to 2-4.

For symmetric and unsymmetric single degree-of-freedom analyses (thermal, etc.), the PCG solver only supports *`Lev_Diff`* = 1.

The **MSAVE (p. 1183)**,ON command is not valid in these circumstances:

- when *`Lev_Diff`* = 5; in this case, the *`Lev_Diff`* value will automatically be reset to 2.
- with the *`StrmCk`* option; in this case, *`StrmCk`* will be set to OFF.
- when the *`Fallback`* option is enabled (set to AUTO or ON); in this case, *`Fallback`* will automatically be reset to OFF.
- when the PCG solver is used for a full harmonic analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Lagrange-formulation contact methods and mixed u-P formulations, the PCG solver cannot be used, and the sparse solver is required.

Fallback logic is automatically disabled (*Fallback* = OFF) in these circumstances:

- for analyses that include MPC184 elements using the Lagrange multiplier method to impose kinematic constraints
- for thermal analyses that use the quasi-static (**THOPT (p. 1997)**,QUASI) option.

Reverting to the PCG Solver After an Automatic Switch to the Sparse Solver

Linear Analysis (symmetric or unsymmetric): When fallback logic is enabled (*Fallback* = AUTO or ON) and the program switches to the sparse direct solver during a linear analysis, the sparse solver is used for the remainder of the simulation unless the solver choice is changed by issuing the **EQSLV (p. 648)** command between load steps.

Nonlinear Analysis (symmetric or unsymmetric): If the program switches to the sparse direct solver during a nonlinear analysis, the sparse solver is used for the remaining equilibrium iterations of the current substep. The program reverts back to the PCG solver at the end of the current substep unless one of the following conditions apply:

- The previous call to the sparse solver involved an indefinite or near-singular matrix.
- The last equilibrium iteration of the current substep using the sparse solver was faster than the last successful equilibrium iteration using the PCG solver.
- Three consecutive fallback switches between the sparse and the PCG solvers have occurred.

For both linear and nonlinear analyses, the solver choice can be changed using the **EQSLV (p. 648)** command between any subsequent load steps.

Full-Harmonic Analysis: If the program switches to the sparse direct solver during a full-harmonic analysis, the sparse solver is used for the current harmonic substep. The program reverts back to the PCG solver at the next continuing substep.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

PCIRC, *RAD1*, *RAD2*, *THETA1*, *THETA2*

Creates a circular area centered about the working plane origin.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RAD1*, *RAD2

Inner and outer radii (either order) of the circle. A value of either zero or blank for either *RAD1* or *RAD2*, or the same value for both *RAD1* and *RAD2*, defines a solid circle.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

THETA1, THETA2

Starting and ending angles (either order) of the circular area. Used for creating a circular sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the *Modeling and Meshing Guide* for an illustration.

Notes

Defines a solid circular area or circular sector centered about the working plane origin. For a solid circle of 360°, the area will be defined with four keypoints and four lines. See the **CYL4 (p. 477)** and **CYL5 (p. 478)** commands for alternate ways to create circles.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>By Dimensions

---

/PCIRCLE, *XCENTR, YCENTR, XLRAD*

Creates an annotation circle (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCENTR

Circle X center location (-1.0 < X < 2.0).

YCENTR

Circle Y center location (-1.0 < Y < 1.0).

XLRAD

Circle radius length.

Notes

Creates an annotation circle to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if annotation is used.

This command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

All circles are shown on subsequent displays unless the annotation is turned off or deleted. Issue **/LSPEC (p. 1081)** and **/PSPEC (p. 1548)** to set the attributes of the circle.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

PCROSS, *LabXR*, *LabYR*, *LabZR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Calculates the cross product of two path vectors along the current path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabXR

Label assigned to X-component of resultant vector.

LabYR

Label assigned to Y-component of resultant vector.

LabZR

Label assigned to Z-component of resultant vector.

LabX1

X-component of first vector label (labeled path item).

LabY1

Y-component of first vector label.

LabZ1

Z-component of first vector label.

LabX2

X-component of second vector label (labeled path item).

LabY2

Y-component of second vector label.

LabZ2

Z-component of second vector label.

Menu Paths

Main Menu>General Postproc>Path Operations>Cross Product

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDEF,*Lab*,*Item*,*Comp*,*Avglab*

Interpolates an item onto a path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Label assigned to the resulting path item (8 characters maximum). This item may be used as input for other path operations.

Item

Label identifying the item for interpolation. Valid item labels are shown in Table 209: PDEF - Valid Item and Component Labels (p. 1373) below. Some items also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in Table 209: PDEF - Valid Item and Component Labels (p. 1373) below.

Avglab

Option to average across element boundaries:

AVG

Average element results across elements (default).

NOAV

Do not average element results across elements. If the parameter *DISCON* = MAT on the **PMAP (p. 1449)** command, this option is automatically invoked.

Notes

Defines and interpolates a labeled path item along a predefined path (**PATH (p. 1354)**). Path item results are in the global Cartesian coordinate directions unless transformed (**RSYS (p. 1639)**). A path item must be defined before it can be used with other path operations. Additional path items may be defined from the **PVECT (p. 1556)**, **PCALC (p. 1362)**, **PDOT (p. 1375)**, and **PCROSS (p. 1371)** commands. Path items may be listed (**PRPATH (p. 1517)**) or displayed (**PLPATH (p. 1439)**, **PLPAGM (p. 1438)**). A maximum number of path items permitted is established by the *nSets* argument specified with the **PATH (p. 1354)** command.

When you create the first path item (**PDEF (p. 1372)** or **PVECT (p. 1556)**), the program automatically interpolates four path items which are used to describe the geometry of the path. These predefined items are the position of the interpolated path points (labels XG, YG, and ZG) in global Cartesian coordinates, and the path length (label S). For alternate methods of mapping the path geometry (to include, for example, material discontinuity) see the **PMAP (p. 1449)** command. These items may also be listed or displayed with the **PRPATH (p. 1517)**, **PLPATH (p. 1439)**, and **PLPAGM (p. 1438)** commands.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If specifying that load case operations act on principal/equivalent stresses (**SUMTYPE (p. 1880)**,PRIN), derived quantities (principal and equivalent stresses/strains) will be zero for path plots. A typical use for such a case involves mode combinations in a response spectrum analysis.

The number of interpolation points on the path is defined by the *nDiv* argument on the **PATH (p. 1354)** command. See Mapping Nodal and Element Data onto the Path in the *Mechanical APDL Theory Reference* for details. Use **PDEF (p. 1372)**,STAT to list the path item labels. Use **PDEF (p. 1372)**,CLEAR to erase all labeled path items, except the path geometry items (XG, YG, ZG, S).

See also Mapping Results onto a Path in the *Basic Analysis Guide*.

Table 209: PDEF - Valid Item and Component Labels

Item	Comp	Description
Valid item and component labels for nodal degree of freedom results are:		
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP[1]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum.
CONC		Concentration.
CURR		Current.
EMF		Electromotive force drop.
Valid item and component labels for element results are:		
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or Equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1, 2, 3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1, 2, 3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
For contact results PowerGraphics is applicable for 3D models only.		
CONT	STAT2	Contact status.
"	PENE	Contact penetration.
"	PRES	Contact pressure.
"	SFRIC	Contact friction stress.
"	STOT	Contact total stress (pressure plus friction).
"	SLIDE	Contact sliding distance.
"	GAP	Contact gap distance.
"	FLUX	Total heat flux at contact surface.
TG	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component pressure gradient or vector sum.
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
JC	X, Y, Z, SUM	Component conduction current density or vector sum (for elements that support conduction current calculation)
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum.
ETAB	Lab	Any user-defined element table label (see **ETABLE (p. 687)** command).
BFE	TEMP	Applied and calculated temperatures along a defined path.
SPL		Sound pressure level.
SPLA		A-weighted sound pressure level (dBA).

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, ..., TTOP instead of TEMP.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For more information on the meaning of contact status and its possible values, see Reviewing Results in POST1 in the *Contact Technology Guide*.

Menu Paths

Main Menu>General Postproc>Path Operations>Clear Path Items

Main Menu>General Postproc>Path Operations>Define Path>Path Status>Current Path

Main Menu>General Postproc>Path Operations>Map onto Path

Main Menu>General Postproc>Path Operations>Map onto Path>FE Results

---

PDOT, *LabR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Calculates the dot product of two path vectors along the current path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to dot product result.

LabX1

X-component of first vector label (labeled path item).

LabY1

Y-component of first vector label (labeled path item).

LabZ1

Z-component of first vector label (labeled path item).

LabX2

X-component of second vector label (labeled path item).

LabY2

Y-component of second vector label (labeled path item).

LabZ2

Z-component of second vector label (labeled path item).

Menu Paths

Main Menu>General Postproc>Path Operations>Dot Product

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PERBC2D, *LOC1*, *LOC2*, *LOCTOL*, *R1*, *R2*, *TOLR*, *OPT*, *PLNOPT*

Generates periodic constraints for 2D planar magnetic field analyses.

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LOC1

Constant coordinate location of the first plane of nodes. For *PLNOPT* = 1 or 2, the constant coordinate location is the global Cartesian coordinate system (**CSYS (p. 441)**,0) location in the X or Y direction respectively. For *PLNOPT* = 0, the location is the angle in the global cylindrical coordinate system (**CSYS (p. 441)**,1).

LOC2

Constant coordinate location of the second plane of nodes. For *PLNOPT* = 1 or 2, the constant coordinate location is the global Cartesian coordinate system (**CSYS (p. 441)**,0) location in the X or Y direction respectively. For *PLNOPT* = 0, the location is the angle (in degrees) in the global cylindrical coordinate system (**CSYS (p. 441)**,1).

LOCTOL

Tolerance on the constant coordinate location for node selection. Defaults to .00001 for *PLNOPT* = 1 or 2 and .001 degrees for *PLNOPT* = 0.

R1

Minimum coordinate location along the second plane of nodes. For *PLNOPT* = 1 or 2, the coordinate location is the global Cartesian coordinate system location in the Y or X direction respectively. For *PLNOPT* = 0, the coordinate location is the radial coordinate value in the global cylindrical coordinate system. Periodic conditions are not applied to nodes at this location.

R2

Maximum coordinate location along the second plane of nodes. For *PLNOPT* = 1 or 2, the coordinate location is the global Cartesian coordinate system location in the Y or X direction respectively. For *PLNOPT* = 0, the coordinate location is the radial coordinate value in the global cylindrical coordinate system. Periodic conditions are not applied to nodes at this location.

TOLR

Tolerance dimension on node selection along the plane of nodes. Defaults to .00001.

OPT

Periodic option:

0

Odd symmetry (default). Apply constraint equations such that AZ(i) = -AZ(j).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Even symmetry. Apply node coupling such that AZ(i) = AZ(j).

PLNOPT

Symmetry plane option:

0

Planes of constant angle in the global cylindrical coordinate system (**CSYS (p. 441)**,1).

1

Planes parallel to the global Cartesian X axis (**CSYS (p. 441)**,0).

2

Planes parallel to the global Cartesian Y axis (**CSYS (p. 441)**,0).

Notes

PERBC2D (p. 1376) invokes a Mechanical APDL macro which generates periodic boundary condition constraints for 2D planar magnetic field analysis.

The macro is restricted to node pairs sharing common coordinate values along symmetry planes separated by a constant coordinate value. Planes (or lines) must lie at either constant angles (*PLNOPT* = 0), constant X values (*PLNOPT* = 1), or constant Y values (*PLNOPT* = 2).

The macro applies constraint equations (*OPT* = 0, odd symmetry) or node coupling (*OPT* = 1, even symmetry) to each node pair sharing a common coordinate value along the symmetry planes. By default, periodic conditions are not applied at the first and last node pairs on the symmetry planes unless the input location values, R1 and R2, are adjusted to be less than or greater than the actual node coordinate values.

Nodes are selected for application of constraints via **NSEL (p. 1266)**, with tolerances on the constant coordinate location (*LOCTOL*) and the coordinate location along the plane (RTOL).

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>VectorPot>Periodic BCs

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>VectorPot>Periodic BCs

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PERTURB, *Type*, *MatKey*, *ContKey*, *LoadControl*

Sets linear perturbation analysis options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Type of linear perturbation analysis to be performed:

STATIC

Perform a linear perturbation static analysis.

MODAL

Perform a linear perturbation modal analysis.

BUCKLE

Perform a linear perturbation eigenvalue buckling analysis.

HARMONIC

Perform a linear perturbation full harmonic analysis.

SUBSTR

Perform a linear perturbation substructure generation pass.

OFF

Do not perform a linear perturbation analysis (default).

MatKey

Key for specifying how the linear perturbation analysis uses material properties, valid for all structural elements except contact elements. For more information, see Linear Perturbation Analysis in the *Mechanical APDL Theory Reference*.

AUTO

The program selects the material properties for the linear perturbation analysis automatically (default). The materials are handled in the following way:

- For pure linear elastic materials used in the base analysis, the same properties are used in the linear perturbation analysis.
- For hyperelastic materials used in the base analysis, the material properties are assumed to be linear elastic in the linear perturbation analysis. The material property data (or material Jacobian) is obtained based on the tangent of the hyperelastic material's constitutive law at the point where restart occurs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For hyperviscoelastic materials used in the base analysis, the program uses the harmonic material formulation in perturbed full harmonic solutions.
- For other nonlinear materials used in the base analysis, the material properties are assumed to be linear elastic in the linear perturbation analysis. The material data is the same as the linear portion of the nonlinear materials (that is, the parts defined via **MP (p. 1160)** commands).
- For COMBIN39, the stiffness is that of the first segment of the force-deflection curve.

TANGENT

Use the tangent (material Jacobian) on the material constitutive curve as the material property. The material property remains linear in the linear perturbation analysis and is obtained at the point of the base analysis where restart occurs. The materials are handled in the following way:

- For pure linear elastic materials used in the base analysis, the same properties are used in the linear perturbation analysis. Because the material constitutive curve is linear, the tangent is the same as the base analysis.
- For hyperelastic materials used in the base analysis, the program uses the same tangent as that used for *MatKey* = AUTO, and the results are therefore identical.
- For hyperviscoelastic materials used in the base analysis, the program uses the harmonic material formulation in perturbed full harmonic solutions.
- For other nonlinear materials used in the base analysis, the material properties are obtained via the material tangent on the material constitutive curve at the restart point of the base analysis.

  The materials and properties typically differ from *Matkey* = AUTO, but it is possible the results could be identical or very similar if a.) the material is elastoplastic rate-independent and is unloading (or has neutral loading) at the restart point, or b.) the material is rate-dependent, depending on the material properties and loading conditions.
- For COMBIN39, the stiffness is equal to the tangent of the current segment of the force-deflection curve.
- In a modal restart solution that follows a linear perturbation modal analysis, the TANGENT option is overridden by the AUTO option and linear material properties are used for stress calculations in the modal restart. See the discussion in the Notes (p. 1382) for more information.

SPOFF

Provide the same values as AUTO, but set the spin softening matrix to zero (ignoring the spin softening effect).

The spin softening effect is excluded in all the linear perturbation analysis types except for linear perturbation buckling. Note that although the spin softening effect is excluded in linear perturbation analysis, it is still included in the base static or full transient analysis if **NLGEOM (p. 1231)**,ON is issued in the base analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ContKey

Key that controls contact status for the linear perturbation analysis. This key controls all contact elements (TARGE169, TARGE170, CONTA172, CONTA174, CONTA175, CONTA177, and CONTA178) globally for all contact pairs. Alternatively, contact status can be controlled locally per contact pair by using the **CNKMOD (p. 392)** command. Note that the contact status from the base analysis solution is always adjusted by the local contact controls specified by **CNKMOD (p. 392)** first and then modified by the global sticking or bonded control (*ContKey* = STICKING or BONDED). The tables in the Notes (p. 1381) section show how the contact status is adjusted by **CNKMOD (p. 392)** and/or the *ContKey* setting.

CURRENT

Use the current contact status from the restart snapshot (default). If the previous run is nonlinear, then the nonlinear contact status at the point of restart is frozen and used throughout the linear perturbation analysis.

STICKING

For frictional contact pairs (MU > 0), use sticking contact (the tangential stiffness is calculated internally based on friction coefficient MU and the element shape) everywhere the contact state is closed (that is, status is sticking or sliding). This option only applies to contact pairs that are in contact and have a frictional coefficient MU greater than zero. Contact pairs without friction (MU = 0) and in a sliding state remain free to slide in the linear perturbation analysis.

BONDED

Any contact pairs that are in the closed (sticking or sliding) state are moved to bonded (for example, KN for both normal and tangential contact stiffness). Contact pairs that have a status of far-field or near-field remain open.

LoadControl

Key that controls how the load vector of $\{F_{perturbed}\}$ is calculated. This control is provided for convenience of load generation for linear perturbation analysis. In general, a new set of loads is required for a linear perturbation analysis. This key controls all mechanical loads; it does not affect non-mechanical loads. Non-mechanical loads (including thermal loads) are always kept (that is, not deleted).

ALLKEEP

Keep all the boundary conditions (loads and constraints) from the end of the load step of the current restart point. This option is convenient for further load application and is useful for a linear perturbation analysis restarted from a previous linear analysis. For this option, $\{F_{end}\}$ is the total load vector at the *end* of the load step at the restart point.

INERKEEP

Delete all loads and constraints from the restart step, except for displacement constraints and inertia loads (default). All displacement constraints (including coupling and constraint equations) and inertia loads are kept for convenience when performing the linear perturb-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ation analysis. Note that nonzero and tabular displacement constraints can be considered as external loads; however, they are not deleted when using this option.

PARKEEP

Delete all loads and constraints from the restart step, except for displacement constraints. All displacement constraints (including coupling and constraint equations) are kept for convenience when performing the linear perturbation analysis. Note that nonzero and tabular displacement constraints can be considered as external loads; however, they are not deleted when using this option.

DZEROKEEP

Behaves the same as the PARKEEP option, except that all nonzero displacement constraints (including any nonzero constant terms defined for any constraint equations) are set to zero upon the onset of linear perturbation.

NOKEEP

Delete all the loads and constraints, including all displacement constraints. This excludes coupling and constraint equations which remain defined, however, all nonzero constant equations are set to zero upon the onset of linear perturbation. For this option, $\{F_{end}\}$ is zero unless non-mechanical loads (for example, thermal loads) are present.

Command Default

Linear perturbation analysis is disabled (*Type* = OFF) by default. When the linear perturbation analysis is enabled, linear material property behavior is assumed for stress calculations; contact status for all contact pairs from the point of restart is used by default; and all loads and constraints from the restart step are deleted, except for displacement constraints and inertia loads, by default.

Notes

This command controls options relating to linear perturbation analyses. It must be issued in the first phase of a linear perturbation analysis.

This command is also valid in PREP7.

Linear Perturbation Overview

A linear perturbation analysis consists of two phases (two **SOLVE (p. 1822)** commands). The first phase is a restart from a base analysis. This base analysis must be a linear or nonlinear static analysis or full transient analysis. The first phase starts with the **ANTYPE (p. 162)**,,RESTART,,,PERTURB command and ends with the **SOLVE (p. 1822)**,ELFORM command. The purpose of the first phase is to re-establish a snapshot of the stiffness matrices at the specified restart point. The second phase, ending with the second **SOLVE (p. 1822)** command, is for the actual linear perturbation analysis.

The total perturbed loads are calculated as follows:

$$\{F_{perturbed}\} = \{F_{end}\} + \{F_{add}\}$$

where:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\{F_{end}\}$ = total loads at the *end* of the load step of the current restart point (load applications are read from the `.ldhi` file). By default, all of the loads in $\{F_{end}\}$ are deleted except for displacement boundary conditions and inertia loads (see the description of *LoadControl* above).

$\{F_{add}\}$ = Additional (new) loads prescribed by the user in the second phase of the linear perturbation analysis (after the first **SOLVE (p. 1822)** command is invoked).

In the first phase of a linear perturbation analysis, the **ANTYPE (p. 162)**,,RESTART command resumes the `Jobname.rdb` database and reads in the `.ldhi` file to establish the $\{F_{end}\}$ load. New load application (adding to $\{F_{add}\}$) or load removal (changing $\{F_{end}\}$) can be done only in the second phase of the linear perturbation analysis (after the first **SOLVE (p. 1822)** command), allowing flexibility in controlling the final $\{F_{perturbed}\}$ to be used.

For *Type* = STATIC, $\{F_{perturbed}\}$ is the actual external load for the static analysis.

For *Type* = MODAL, $\{F_{perturbed}\}$ is calculated and stored in the `.full` and `.mode` files for a subsequent mode-superposition, PSD, or other type of modal-based linear dynamic analysis. Linear dynamic options such as multiple load generations (**MODCONT (p. 1132)**,ON), enforced motion (**MODCONT (p. 1132)**, ,ON), and residual vector methods (**RESVEC (p. 1603)**,ON) can be used in a linear perturbation analysis. For these methods, the **MODCONT (p. 1132)** or **RESVEC (p. 1603)** command must be invoked in the second phase (after the first **SOLVE (p. 1822)**) of the linear perturbation procedure. For the enforced motion method, the base identification number should be specified (**D (p. 483)** command) in the second phase of the linear perturbation analysis. This base identification number is used later in the downstream mode-superposition or other mode-superposition based analysis.

For *Type* = BUCKLE, $\{F_{perturbed}\}$ is the actual linear buckling load which is used to generate the linear stress stiffening matrix for the buckling analysis.

For *Type* = HARMONIC, $\{F_{perturbed}\}$ is the actual external load for the full harmonic analysis. In this case, $\{F_{perturbed}\}$ can be frequency dependent and can use complex input.

For *Type* = SUBSTR, $\{F_{perturbed}\}$ is used to generate the first reduced external load vector of the sub-structure.

Controlling Material Behavior

In most cases involving linear perturbation analysis, *Matkey* = AUTO is the best option for controlling material behavior. *Matkey* = TANGENT is often the better option, however, in special cases such as the following:

- A linear perturbation buckling analysis, to introduce preferred buckling modes into a subsequent post-buckling nonlinear analysis.
- A linear perturbation modal analysis, to introduce preferred modes into a subsequent bifurcation analysis.

If the TANGENT option is used in conjunction with a modal restart solution that follows a linear perturbation modal analysis, then the AUTO option is assumed and linear material properties are used for stress calculations in the modal restart solution. This occurs because the TANGENT material properties are not available during the modal restart phase due to a data architecture limitation. Furthermore, linear ma-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

terial properties are used for the stress calculation in any downstream analysis that uses the modal restart solution.

For more information about the automatic and tangent options, see Specifying Material Behavior in Linear Perturbation in the *Element Reference*.

Controlling Contact Status

You can control the contact status for the linear perturbation analysis by using the *ContKey* field on this command and/or the **CNKMOD (p. 392)** command. The first table shows the effects of using only the *ContKey* setting on the **PERTURB (p. 1378)** command. The second table shows the effects of using both the **CNKMOD (p. 392)** command and the *ContKey* setting on **PERTURB (p. 1378)**.

Table 210: Adjusted Contact Status when PERTURB Command Is Issued

Contact Status from the Base Analysis Solution at the Restart Point	***ContKey* Setting on PERTURB Command**	
	ContKey Value	Adjusted Contact Status
0 - far-field	any	0 - far-field
1 - near-field	any	1 - near-field
2 - sliding	CURRENT or STICKING (mu=0)	2 - sliding
	STICKING (mu>0) or BONDED	3 - sticking
3 - sticking	any	3 - sticking

Table 211: Adjusted Contact Status when Both CNKMOD and PERTURB Are Issued

Contact Status from the Base Analysis Solution at the Restart Point	**CNKMOD,*ITYPE*,12,*Value***		***ContKey* Setting on PERTURB Command**	
	KEYOPT(12) Value	Adjusted Contact Status	*ContKey* Value	Final Adjusted Contact Status
0 - far-field	any	0 - far-field	any	0 - far-field
1 - near-field	0, 1, 2, 3, 6	1 - near-field	any	1 - near-field
	4	1 - near-field (if outside of the adjusted pinball region)	any	1 - near-field
		2 - sliding (if inside of the adjusted pinball region)	CURRENT or STICKING (mu=0)	2 - sliding
			STICKING (mu>0) or BONDED	3 - sticking

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	5	1 - near-field (if outside of the adjusted pinball region)	any	1 - near-field
		3 - sticking (if inside of the adjusted pinball region)	any	3 - sticking
2 - sliding	0, 2, 4	2 - sliding	CURRENT or STICKING (mu=0)	2 - sliding
			STICKING (mu>0) or BONDED	3 - sticking
	1, 3, 5, 6	3 - sticking	any	3 - sticking
3 - sticking	any	3 - sticking	any	3 - sticking

When *ContKey* is set to CURRENT, all contact related parameters (such as normal stiffness and tangential stiffness) will remain unchanged throughout the linear perturbation analysis. However when *ContKey* is set to STICKING or BONDED, the program will re-evaluate the contact normal and tangential stiffness in order to perform the linear perturbation analysis based on the actual sticking behavior regardless of the friction coefficient value.

Note that the **CNKMOD (p. 392)** command allows you to take points in the base analysis that are near contact (within the pinball region) and modify them to be treated as "in contact" in the linear perturbation analysis; see the "1 - near-field" row in the above table with KEYOPT(12) values set to 4 or 5. **CNKMOD (p. 392)** also allows you to take points that are sliding in the base analysis and treat them as sticking in the linear perturbation analysis, irrespective of the MU value; see the "2 - sliding" row in the above table with KEYOPT(12) values set to 1,5, or 6.

If an open gap exists at the restart point of the base static/transient solution and the contact status is adjusted as sliding or sticking due to a “bonded” or “no separation” contact behavior definition, then the program will treat it as near-field contact when executing the **CNKMOD (p. 392)** command in a downstream linear perturbation analysis.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Restart

Main Menu>Solution>Analysis Type>Restart

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PFACT, *TBLNO*, *Excit*, *Parcor*

Calculates participation factors for the PSD or multi-point response spectrum table.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Input PSD (Power Spectral Density) table number for which participation factors are to be calculated.

Excit

Label defining the location of excitation:

BASE

Base excitation (default).

NODE

Nodal excitation.

Parcor

Label defining excitation type (applies only to **SPOPT (p. 1837)**,PSD analysis). Used only when partially correlated excitation is due to wave propagation or spatial correlation. Defaults to partially correlated excitation as defined by **COVAL (p. 420)** and **QDVAL (p. 1559)** commands.

WAVE

Excitation defined by **PSDWAV (p. 1534)** command.

SPAT

Excitation defined by **PSDSPL (p. 1532)** command.

Notes

Calculates the participation factors for a particular PSD or multi-point response spectrum table defined with the **PSDVAL (p. 1534)** or **SPVAL (p. 1842)** command. The `Jobname.db` file must contain modal solution data in order for this command to calculate the participation factor. There must be a **PFACT (p. 1385)** command for each excitation spectrum. You are limited to 300 excitations.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Calculate PF

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Calculate PF

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Calculate PF

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Calculate PF

PIVCHECK, *KEY*, *PRNTCNTRL*

Controls the behavior of an analysis when a negative or zero equation solver pivot value is encountered.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Determines whether to stop or continue an analysis when a negative or zero equation solver pivot value is encountered:

AUTO

Check for negative or zero pivot values for analyses performed with the sparse and PCG solvers. When one is encountered, an error or warning is issued, per various criteria relating to the type of analysis being solved. An error causes the analysis to stop; a warning allows the analysis to continue. A negative pivot value may be valid for some nonlinear and multiphysics analyses (for example, electromagnetic and thermal analyses); this key has no effect in these cases.

ERROR

Check for negative or zero pivot values for analyses performed with the sparse and PCG solvers. When one is encountered, an error is issued, stopping the analysis. A negative pivot value may be valid for some nonlinear and multiphysics analyses (for example, electromagnetic and thermal analyses); this key has no effect in these cases.

WARN

Check for negative or zero pivot values for analyses performed with the sparse and PCG solvers. When one is encountered, a warning is issued and the analysis continues. A negative pivot value may be valid for some nonlinear and multiphysics analyses (for example, electromagnetic and thermal analyses); this key has no effect in these cases.

OFF

Pivot values are not checked. This key causes the analysis to continue in spite of a negative or zero pivot value.

PRNTCNTRL

Provides print options. Print output with these options will be sent to the default output file, not to the files created by the nonlinear diagnostic tools (**NLDIAG (p. 1224)**).

ONCE

Print only the maximum and minimum pivot information on the first call to the sparse solver (which is the default solver). This is the default behavior.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EVERY

Print the maximum and minimum pivot information at every call to the sparse solver. This option is provided for nonlinear analysis diagnostics.

Command Default

The program checks for negative or zero pivot values (*Key* = AUTO). If any are found, the analysis may stop with an error or may proceed with only a warning, depending on various criteria pertaining to the type of analysis being solved.

Notes

This command is valid for all analyses. In a nonlinear analysis, a negative pivot may be valid. In some cases, rigid body motions in a nonlinear analysis will be trapped by error routines checking infinitely large displacements (DOF limit exceeded) or nonconvergence status. An under-constrained model may avoid the pivot check, but fail with a DOF limit exceeded error.

Machine precision may affect whether a small pivot triggers an error or bypasses this checking logic. You may wish to review the ratio of the maximum to absolute minimum pivot values. For ratios exceeding 12 to 14 orders of magnitude, the accuracy of the computed solution may be degraded by the severe ill-conditioning of the assembled matrix.

Note that negative pivots corresponding to Lagrange multiplier based mixed u-P elements are not checked or reported by this command. Negative pivots arising from the u-P element formulation and related analyses can occur and lead to correct solutions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

PLAS, *Lab*, *LDSTEP*, *SUBSTEP*, *FREQB*, *FREQE*, *LogOpt*, *PlotType*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*

Plots a specified acoustic quantity during postprocessing of an acoustic analysis.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

The acoustic quantity to calculate:

SIMP

Specific acoustic impedance on the selected surface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AIMP

Acoustic impedance on the selected surface.

MIMP

Mechanical impedance on the selected surface.

PRES

Average pressure on the selected surface.

FORC

Force on the selected surface.

POWE

Acoustic power on the selected surface.

ERP

Equivalent radiated power on the selected structural surface (valid only for SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, SOLID225, SOLID226, SOLID227, and SHELL281).

ERPL

Equivalent radiated power level on the selected structural surface (valid only for SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, SOLID225, SOLID226, SOLID227, and SHELL281).

BSPL

Frequency-band sound pressure level on selected nodes.

BSPA

A-weighted frequency-band sound pressure level on selected nodes.

MENE

Acoustic potential energy on the selected elements.

KENE

Acoustic kinetic energy on the selected elements.

TENE

Acoustic total energy on the selected elements.

PL2V

Average square of the L2 norm of pressure on the selected elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LWIN

Input sound power level on defined port.

LWOUT

Output sound power level on defined driven port.

RL

Return loss on defined port.

ALPHA

Absorption coefficient on defined port.

TL

Transmission loss on defined ports.

DFSTL

Transmission loss of random acoustic analysis.

DFSPW

Radiated power in random acoustic analysis.

LDSTEP

Specified load step. Defaults to the load step number specified on the **SET (p. 1724)** command, or defaults to 1 if **SET (p. 1724)** has not been issued. This default applies to all *Lab* values except DFSTL and DFSPW.

n

Load step number.

ALL

All load steps.

AVG or 0

Average result of multiple samplings in a random acoustic analysis (see the **MSOLVE (p. 1193)** command). This option is used only for *Lab* = DFSTL and DFSPW, and it is the default for these labels.

SUBSTEP

Specified substep. Defaults to the substep number specified on the **SET (p. 1724)** command, or defaults to ALL (all substeps at the specified load step) if **SET (p. 1724)** has not been issued. For *Lab* = BSPL or BSPA, ALL is the only valid value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

n

Substep number.

ALL

All substeps.

FREQB

Frequency value representing one of the following:

- Beginning frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If a *SUBSTEP* value is specified, *FREQB* is invalid.
- Central frequency of octave bands, used when *LogOpt* = OB1, OB2, OB3, OB6, OB12, or OB24 and *FREQE* is blank.

FREQE

Ending frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If blank, *FREQE* is set to *FREQB*. If a *SUBSTEP* value is specified, *FREQE* is invalid.

LogOpt

Octave bands:

OB0

Narrow bands (default).

OB1

Octave bands.

OB2

1/2 octave bands.

OB3

1/3 octave bands.

OB6

1/6 octave bands.

OB12

1/12 octave bands.

OB24

1/24 octave bands.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PlotType

Type of plot:

LINE

Line plot (default).

BAR

Bar pattern plot, used only for *Lab* = BSPL or BSPA.

CONT

Waterfall diagram, used only for *Lab* = ERP or ERPL.

VAL1

Input port number for *Lab* = LWIN, LWOUT, RL, ALPHA, or TL.

VAL2

Output port number for *Lab* = TL.

VAL3

Reference power for *Lab* = LWIN, LWOUT, or EPRL (defaults to $1x10^{-12}$ W).

VAL4

Fluid mass density for *Lab* = ERP or ERPL (defaults to 1.2041 kg/m^3).

VAL5

Speed of sound in the fluid for *Lab* = ERP or ERPL (defaults to 343.25 m/s).

VAL6

Radiation efficiency for *Lab* = ERP or ERPL (defaults to 1).

Notes

The **PLAS (p. 1387)** command plots the specified acoustic quantity on the selected exterior surface, the energy on selected elements, or the sound pressure level over frequency bands. The calculation is based on the pressure and velocity solution or the frequency-band sound pressure level (SPL).

The total pressure and velocity are used if the selected surface is the excitation source surface. To calculate the incoming and outgoing acoustic power and other sound power parameters on the input and output surfaces, issue the **SF (p. 1733)**,,PORT command in the preprocessor to define port numbers.

The sound pressure level of the octave bands and general frequency band (defined via the **HARFRQ (p. 885)** command) is calculated at the selected nodes in the model.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

PLCAMP,*Option*,*SLOPE*,*UNIT*,*FREQB*,*Cname*,*STABVAL*,*KeyAllFreq*,*KeyNegFreq*

Plots Campbell diagram data for applications involving rotating structure dynamics.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Flag to activate or deactivate sorting of forward or backward whirl frequencies:

0 (OFF *or* NO)

No sorting.

1 (ON *or* YES)

Sort. This value is the default.

SLOPE

The slope of the line to be printed. This value must be positive.

SLOPE > 0

In the stationary reference frame (*RefFrame* = YES on the **CORIOLIS (p. 417)** command), the line represents the number of excitations per revolution of the rotor. For example, *SLOPE* = 1 represents one excitation per revolution, usually resulting from unbalance.

In the rotating reference frame (*RefFrame* = NO on the **CORIOLIS (p. 417)** command), the line represents the number of excitations per revolution of the rotor minus 1.

SLOPE = 0

The line represents the stability threshold for stability values or logarithmic decrements printout (*STABVAL* = 1, 2, or 3)

UNIT

Specifies the unit of measurement for rotational angular velocities:

RDS

Rotational angular velocities in radians per second (rad/s). This value is the default.

RPM

Rotational angular velocities in revolutions per minute (RPMs).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FREQB

The beginning, or lower end, of the frequency range of interest. The default is zero.

Cname

The rotating component name.

STABVAL

Flag to plot the stability values:

0 (OFF *or* NO)

Plot the frequencies (the imaginary parts of the eigenvalues in Hz). This value is the default.

1 (ON *or* YES)

Plot the stability values (the real parts of the eigenvalues in Hz).

2

Plot the inverse of the logarithmic decrements. A negative logarithmic decrement indicates stable motion.

3

Plot the logarithmic decrements. A positive logarithmic decrement indicates stable motion and is consistent with API (American Petroleum Institute) standards.

For more information about complex eigenmodes and corresponding logarithmic decrements, see Complex Eigensolutions in the *Mechanical APDL Theory Reference*.

KeyAllFreq

Key to specify if all frequencies above FREQB are plotted:

0 (OFF *or* NO)

A maximum of 10 frequencies are plotted. This value is the default.

1 (ON *or* YES)

All frequencies are plotted.

KeyNegFreq

Key to specify if the negative frequencies are plotted. It only applies to solutions obtained with the damped eigensolver (*Method* = DAMP on the **MODOPT (p. 1140)** command):

0 (OFF *or* NO)

Only positive frequencies are plotted. This value is the default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 (ON *or* YES)

Negative and positive frequencies are plotted.

Notes

The following items are required when generating a Campbell diagram:

- Activate the Coriolis effect (**CORIOLIS (p. 417)** command) in the solution phase (**/SOLU (p. 1821)**).
- Run a modal analysis using the QR damped (**MODOPT (p. 1140)**,QRDAMP) or damped (**MODOPT (p. 1140)**,DAMP) method. Complex eigenmodes are necessary (**MODOPT (p. 1140)**,QRDAMP,,,,*Cpxmod* = ON), and you must specify the number of modes to expand (**MXPAND (p. 1203)**).
- Define two or more load step results with an ascending order of rotational velocity (**OMEGA (p. 1324)** or **CMOMEGA (p. 368)**).

In some cases where modes are not in the same order from one load step to the other, sorting the frequencies (*Option* = 1) can help to obtain a correct plot. Sorting is based on the comparison between complex mode shapes calculated at two successive load steps.

At each load step, the application compares the mode shape to the loads at other load steps to determine whirl direction at the load step. If applicable, a label appears (in the plot legend) representing each whirl mode (BW for backward whirl and FW for forward whirl).

At each load step, the program checks for instability (based on the sign of the real part of the eigenvalue). The labels "stable" or "unstable" appear in the plot legend for each frequency curve.

The rotational velocities of a named component (*Cname*) are displayed on the X-axis.

For information on plotting a Campbell diagram for a prestressed structure, see Solving for a Subsequent Campbell Analysis of a Prestressed Structure Using the Linear Perturbation Procedure in the *Rotordynamic Analysis Guide*.

For a usage example of the **PLCAMP (p. 1392)** command, see Campbell Diagram in the *Rotordynamic Analysis Guide*.

Damped modal cyclic symmetry (**CYCLIC (p. 462)**) analyses do not support the **PLCAMP (p. 1392)** command.

Menu Paths

Main Menu>General Postproc>Rotor Dynamics>Plot Campbell

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLCFREQ,*SPEC*,*SECTbeg*,*SECTend*

Plots the frequency response for the given CYCSPEC (p. 475) specification.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SPEC

CYCSPEC (p. 475) specification number (ordered 1 to N in the order input; use **CYCSPEC (p. 475)**,LIST to view the current list order). Defaults to 1.

SECTbeg

Beginning sector number to plot. Defaults to 1.

SECTend

Ending sector number to plot. Defaults to the total number of sectors expanded (**/CYCEXPAND (p. 454)**).

Notes

Following a cyclic mode-superposition harmonic analysis, this command plots the result item given by a **CYCSPEC (p. 475)** specification versus the harmonic frequency, one curve for each of the specified sectors. A **CYCCALC (p. 453)** command must have been issued prior to this command.

Menu Paths

This command cannot be accessed from a menu.

PLCHIST,*SPEC*,*FREQpt*

Plots a histogram of the frequency response of each sector for the given CYCSPEC (p. 475) specification.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SPEC

CYCSPEC (p. 475) specification number (ordered 1 to N in the order input; use **CYCSPEC (p. 475)**,LIST to view the current list order). Defaults to 1.

FREQpt

Harmonic frequency point to plot (the data set number NSET or CUMULATIVE on **SET (p. 1724)**,LIST). Defaults to the current **SET (p. 1724)** frequency.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Following a cyclic mode-superposition harmonic analysis, this command creates a histogram plot of the result item given by a **CYCSPEC (p. 475)** specification versus the sector number. A **CYCCALC (p. 453)** command must have been issued prior to this command.

Menu Paths

This command cannot be accessed from a menu.

PLCINT,*ACTION*,*ID*,*Node*,*Cont*,*Dtype*

Plots the fracture parameter (CINT (p. 334)) result data.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

PATH

Plots **CINT (p. 334)** quantities according to path number (default).

FRONT

Plots **CINT (p. 334)** quantities distribution along the crack front.

ID

Crack ID number.

Node

Crack tip node number (default = ALL).

Use only for *ACTION* = PATH. Plots **CINT (p. 334)** contour for an individual crack tip node.

Cont

Contour number (Default = ALL).

Use only for *ACTION* = FRONT. Plots **CINT (p. 334)** distribution along the crack for a given path.

Dtype

Data type to output:

JINT

J-integral (default)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IIN1

Interaction integral 1

IIN2

Interaction integral 2

IIN3

Interaction integral 3

K1

Mode 1 stress-intensity factor

K2

Mode 2 stress-intensity factor

K3

Mode 3 stress-intensity factor

G1

Mode 1 energy release rate

G2

Mode 2 energy release rate

G3

Mode 3 energy release rate

GT

Total energy release rate

MFTX

Total material force X

MFTY

Total material force Y

MFTZ

Total material force Z

TSTRESS

T-stress

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CEXT

Crack extension

CSTAR

C*-integral

DLTA

Incremental crack extension in a fatigue/static crack-growth analysis

DLTN

Number of incremental cycles in a fatigue crack-growth analysis

DLTK

Equivalent stress-intensity factor range in a fatigue crack-growth analysis

KEQV

Equivalent stress-intensity factor in a fatigue/static crack-growth analysis

KANG

Kink angle (degree) in a fatigue/static crack-growth analysis

R

Stress (load) ratio in a fatigue crack-growth analysis

UFAC

U-factor (crack closure) in a fatigue crack-growth analysis

Notes

The **PLCINT (p. 1396)** command is not available for XFEM-based crack-growth analyses results processing.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLCKSURF, *MODELDISPLAY*

Plots the $\Phi = 0$ level set surface in an XFEM-based crack analysis

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MODELDISPLAY

Solid model display behavior:

0

No display of the solid model (default).

1

Solid model displayed with translucency and edges disabled.

Notes

The **PLCKSURF (p. 1399)** command is available only for XFEM-based crack analysis during results processing.

Menu Paths

This command cannot be accessed from a menu.

PLCPLX, *KEY*

Specifies the part of a complex variable to display.

POST26 (p. 58): Display (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Complex variable part:

0

Amplitude.

1

Phase angle.

2

Real part.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3

Imaginary part.

Notes

Used only with harmonic analyses (**ANTYPE (p. 162)**,HARMIC).

All results data are stored in the form of real and imaginary components and converted to amplitude and/or phase angle as specified via the **PLCPLX (p. 1399)** command. The conversion is not valid for derived results (such as principal stress/strain, equivalent stress/strain and USUM).

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

PLDISP, *KUND*

Displays the displaced structure.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KUND

Undisplaced shape key:

0

Display only displaced structure.

1

Overlay displaced display with similar undisplaced display (appearance is system-dependent).

2

Same as 1 except overlay with undisplaced edge display (appearance is system-dependent).

Notes

Displays the displaced structure for the selected elements.

For information on true scale plots, refer to the description of the **/DSCALE (p. 575)** command (**/DSCALE (p. 575)**,,1.0).

Menu Paths

Main Menu>General Postproc>Plot Results>Deformed Shape

Utility Menu>Plot>Results>Deformed Shape

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Utility Menu>PlotCtrls>Animate>Deformed Shape

PLESOL, *Item*, *Comp*, *KUND*, *Fact*, *AVG*

Displays solution results as discontinuous element contours.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label. For selected result output, specify SRES and see Table 213: PLESOL - Selected Result (SRES) Component Labels (p. 1411).

Comp

Component of the item (if required). Valid component labels are shown in the table below.

KUND

Undisplaced shape key:

0

Do not overlay undeformed structure display.

1

Overlay displaced contour plot with undeformed display (appearance is system-dependent).

2

Overlay displaced contour plot with undeformed edge display (appearance is system-dependent).

Fact

Scale factor for 2D display of contact items. Default = 1. To invert the display, specify a negative scaling factor.

AVG

Specifies whether results of reinforcing members within the same reinforcing element are smoothed:

0

Disable smoothing.

1

Enable smoothing (default), displaying constant results of reinforcing members if the base elements are low-order, and linear results when the base elements are high-order.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

PLESOL (p. 1401) displays the solution results as element contours discontinuous across element boundaries for the selected elements.

For example, **PLESOL (p. 1401)**,S,X displays the X component of stress S (that is, the SX stress component). Various element results depend on the calculation method and the selected results location (**AVPRIN (p. 210)**, **RSYS (p. 1639)**, and **ESEL (p. 661)**).

Contours are determined by linear interpolation within each element, unaffected by the surrounding elements; that is, no nodal averaging occurs. The discontinuity between contours of adjacent elements is an indication of the gradient across elements. Component results are displayed in the active results coordinate system (**RSYS (p. 1639)** [default is global Cartesian]).

To display items not available via **PLESOL (p. 1401)** (such as line element results), see **ETABLE (p. 687)** and **PLETAB (p. 1411)**.

For PowerGraphics displays (**/GRAPHICS (p. 866)**,POWER), results are plotted only for the model exterior surface. Items not supported by PowerGraphics are noted in Table 212: PLESOL - General Result Item and Component Labels (p. 1402).

The results displayed by **PLESOL (p. 1401)** are unaffected by any requested nodal-averaged results (**OUTRES (p. 1336)**,NAR). For more information, see Nodal-Averaged Results in the *Element Reference*.

For *Item* = SRES, selected result (**OSRESULT (p. 1327)**) values are output. See Table 213: PLESOL - Selected Result (SRES) Component Labels (p. 1411).

Table 212: PLESOL - General Result Item and Component Labels

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
S	X, Y, Z, XY, YZ, XZ	Component stress.
	1, 2, 3	Principal stress.
	INT	Stress intensity.
	EQV	Equivalent stress.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
	1, 2, 3	Principal elastic strain.
	INT	Elastic strain intensity.
	EQV	Elastic equivalent strain.
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain.[a]
	1, 2, 3	Principal diffusion strain.
	EQV	Diffusion equivalent strain.
	INT	Diffusion strain intensity.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
	1, 2, 3	Principal plastic strain.
	INT	Plastic strain intensity.
	EQV	Plastic equivalent strain.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
	1, 2, 3	Principal creep strain.
	INT	Creep strain intensity.
	EQV	Creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
	1, 2, 3	Principal thermal strain.
	INT	Thermal strain intensity.
	EQV	Thermal equivalent strain.
EPSW		Swelling strain.
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR).
	1, 2, 3	Principal total mechanical strain.
	INT	Total mechanical strain intensity.
	EQV	Total mechanical equivalent strain.
EPTT	X, Y, Z, XY, YZ, XZ	Total mechanical, thermal, diffusion, and swelling strain (EPEL + EPPL + EPCR + EPTH + EPDI + EPSW).
	1, 2, 3	Principal total mechanical, thermal, diffusion, and swelling strain.
	INT	Total mechanical, thermal, diffusion, and swelling strain intensity.
	EQV	Total mechanical, thermal, diffusion, and swelling equivalent strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
	SRAT	Stress state ratio.
	HPRES	Hydrostatic pressure.
	EPEQ	Accumulated equivalent plastic strain.
	CREQ	Accumulated equivalent creep strain.
	PSV	Plastic state variable.
	PLWK	Plastic work/volume.
SEND	ELASTIC[b]	Elastic strain energy density. (For viscoelastic and sintering materials, the stored energy.)
	PLASTIC	Plastic strain energy density.
	CREEP	Creep strain energy density.
	DAMAGE	Damage strain energy density.
	VDAM[b]	Viscoelastic dissipation energy density.
	VREG	Visco-regularization strain energy density.
	DISS	Structural-thermal dissipation.
	ENTO	Total strain energy density (sum of ELASTIC, PLASTIC, and CREEP strain energy densities).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
CDM	DMG	Damage variable.
	LM	Maximum previous strain energy for virgin material.
FAIL	MAX	Maximum of all active failure criteria defined at the current location.[a][c]
	EMAX	Maximum Strain Failure Criterion.[a][c]
	SMAX	Maximum Stress Failure Criterion.[a][c]
	TWSI	Tsai-Wu Strength Index Failure Criterion.[a][c]
	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion[a][c]
	HFIB	Hashin Fiber Failure Criterion.[a][c][d]
	HMAT	Hashin Matrix Failure Criterion.[a][c][d]
	PFIB	Puck Fiber Failure Criterion.[a][c][d]
	PMAT	Puck Matrix Failure Criterion.[a][c][d]
	L3FB	LaRc03 Fiber Failure Criterion.[a][c][d]
	L3MT	LaRc03 Matrix Failure Criterion.[c][d]
	L4FB	LaRc04 Fiber Failure Criterion.[a][c][d]
	L4MT	LaRc04 Matrix Failure Criterion.[a][c][d]
	USR1, USR2, ..., USR9	User-defined failure criteria.[a][c][d][e]
PFC	MAX[f]	Maximum of all failure criteria defined at current location.
	FT[f]	Fiber tensile failure criteria.
	FC[f]	Fiber compressive failure criteria.
	MT[f]	Matrix tensile failure criteria.
	MC[f]	Matrix compressive failure criteria.
PDMG	STAT	Damage status (0 = undamaged, 1 = damaged, 2 = completely damaged).
	FT	Fiber tensile damage variable.
	FC	Fiber compressive damage variable.
	MT	Matrix tensile damage variable.
	MC	Matrix compressive damage variable.
	S	Shear damage variable (S).
	SED	Energy dissipated per unit volume.
	SEDV	Energy per unit volume due to viscous damping.
FCMX	LAY	Layer number where the maximum of all active failure criteria over the entire element occurs.[a][c]
	FC	Number of the maximum-failure criterion over the entire element:[a][c] 1 = EMAX 2 = SMAX

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
		3 = TWSI 4 = TWSR 5 = PFIB 6 = PMAT 7 = HFIB 8 = HMAT 9 = L3FB 10 = L3MT 11 = L4FB 12 = L4MT 13~21 = USR1~USR9
	VAL	Value of the maximum failure criterion over the entire element:[a][c]
SVAR	1, 2, 3, ... N	State variable.[a]
GKS	X, XY, XZ	Gasket component stress.
GKD	X, XY, XZ	Gasket component total closure.
GKDI	X, XY, XZ	Gasket component total inelastic closure.
GKTH	X, XY, XZ	Gasket component thermal closure.
SS	X, XY, XZ	Interface traction (stress).
SD	X, XY, XZ	Interface separation.
CONT	STAT	Contact status:[g] 3 = closed and sticking 2 = closed and sliding 1 = open but near contact 0 = open and not near contact
	PENE	Contact penetration.
	PRES	Contact pressure.
	SFRIC	Contact friction stress.
	STOT	Contact total stress (pressure plus friction).
	SLIDE	Contact sliding distance.
	GAP	Contact gap distance.
	FLUX	Total heat flux at contact surface.
	CNOS	Total number of contact status changes during substep.
	FPRS	Fluid penetration pressure.
TG[h]	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF[h]	X, Y, Z, SUM	Component thermal flux or vector sum.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
PG	X, Y, Z, SUM	Component or vector sum of velocity or energy density flux (room acoustics).
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum.[a]
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum.[a]
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum.[a]
P	X, Y, Z, SUM	Poynting vector component or sum.[a]
SERR[i]		Structural error energy.[a]
SDSG[i]		Absolute value of the maximum variation of any nodal stress component.[a]
TERR[i]		Thermal error energy.[a]
TDSG[i]		Absolute value of the maximum variation of any nodal thermal gradient component.[a]
F	X, Y, Z	X, Y, or Z structural force.[a][j]
M	X, Y, Z	X, Y, or Z structural moment.[a]
HEAT		Heat flow.[a]
FLOW		Fluid flow.[a]
AMPS		Current flow.[a] Use **FORCE (p. 772)** for type.
CHRG		Charge.[a] Use **FORCE (p. 772)** for type.
FLUX		Magnetic flux.[a]
CSG	X, Y, Z	X, Y, or Z magnetic current segment component.[a]
RATE		Diffusion flow rate.
SENE		"Stiffness" energy or thermal heat dissipation.[a] Same as TENE.
STEN		Elemental energy dissipation due to stabilization.[a]
TENE		Thermal heat dissipation or "stiffness" energy.[a] Same as SENE.
KENE		Kinetic energy.[a]
ASENE		Amplitude "stiffness" energy.[a]
PSENE		Peak "stiffness" energy.[a]
AKENE		Amplitude kinetic energy.[a]
PKENE		Peak kinetic energy.[a]
DENE		Damping energy.[a]
WEXT[k]		Work due to external load.[a]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
AENE		Artificial energy due to hourglass control/drill stiffness or due to contact stabilization.[a]
JHEAT		Element Joule heat generation.[a]
JS	X, Y, Z, SUM	Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM).[a]
JT	X, Y, Z, SUM	Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM).[a]
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).[a]
MRE		Magnetic Reynolds number.[a]
VOLU		Volume of volume element.[a]
CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system.[a]
BFE	TEMP[l]	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).[m]
SMISC	*snum*	Element summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description.
NMISC	*snum*	Element non-summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description.
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; I1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
FICT	TEMP	Fictive temperature.
ESIG	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
	1, 2, 3	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
	GPOR	Porosity due to void growth.
	NPOR	Porosity due to void nucleation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
FFLX	X,Y,Z	Fluid flow flux in poromechanics.
FGRA	X,Y,Z	Fluid pore-pressure gradient in poromechanics.
MENE		Acoustic potential energy.[a]
PMSV	VRAT, PPRE, DSAT, RPER	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure-thermal elements.
YSIDX	TENS,SHEA	Yield surface activity status for Mohr-Coulomb, soil, concrete, and joint rock material models: 1 = yielded, 0 = not yielded.
FPIDX	TF01,SF01, TF02,SF02, TF03,SF03, TF04,SF04	Failure plane surface activity status for concrete and joint rock material models: 1 = yielded, 0 = not yielded. Tension and shear failure status are available for all four sets of failure planes.
NS	X, Y, Z, XY, YZ, XZ	Nominal strain for hyperelastic material, reported in the current configuration (unaffected by **RSYS (p. 1639)**).
MPLA	DMAC, DMAX	Microplane damage, macroscopic and maximum values.
MPDP	TOTA, TENS, COMP, RW	Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).
DAMAGE	1,2,3,MAX	Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).
GDMG		Damage
IDIS		Structural-thermal dissipation rate
BKS	X, Y, Z, XY, YZ, XZ	Total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements.
BKS1, . . . ,BKS5	X, Y, Z, XY, YZ, XZ	Superimposed components of the total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements when more than one superimposed back-stress component is defined.
EPFR		Free strain in porous media
SNDI	X, Y, Z, SUM	Component sound intensity or vector sum.[a]
FC1S	1,2,3,4,5,6	First set of six components of FCC crystal slip. Available for 3D elements only.
FC2S	1,2,3,4,5,6	Second set of six components of FCC crystal slip. Available for 3D elements only.
HC1S	1,2,3,4,5,6	Six components of HCP crystal slip on basal and prismatic systems. Available for 3D elements only.
HC2S	1,2,3,4,5,6	Six components of HCP crystal slip on pyramidal system. Available for 3D elements only.
HC3S	1,2,3,4,5,6	First set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
HC4S	1,2,3,4,5,6	Second set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC5S	1,2,3,4,5,6	Six components of HCP crystal slip on the second-order pyramidal system. Available for 3D elements only.
BC1S	1,2,3,4,5,6	First set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC2S	1,2,3,4,5,6	Second set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC3S	1,2,3,4,5,6	First set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC4S	1,2,3,4,5,6	Second set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC5S	1,2,3,4,5,6	First set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC6S	1,2,3,4,5,6	Second set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC7S	1,2,3,4,5,6	Third set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC8S	1,2,3,4,5,6	Fourth set of six components of BCC slip on 123 plane. Available for 3D elements only.
FC1H	1,2,3,4,5,6	First set of six components of FCC crystal hardness. Available for 3D elements only.
FC2H	1,2,3,4,5,6	Second set of six components of FCC crystal hardness. Available for 3D elements only.
HC1H	1,2,3,4,5,6	Sixcomponents of HCP crystal hardness on basal and prismatic systems. Available for 3D elements.
HC2H	1,2,3,4,5,6	Six components of HCP crystal hardness on pyramidal system. Available for 3D elements only.
HC3H	1,2,3,4,5,6	First set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC4H	1,2,3,4,5,6	Second set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC5H	1,2,3,4,5,6	Six components of HCP crystal hardness on the second-order pyramidal system. Available for 3D elements only.
BC1H	1,2,3,4,5,6	First set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC2H	1,2,3,4,5,6	Second set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC3H	1,2,3,4,5,6	First set of six components of BCC hardness on 112 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLESOL (p. 1401), *Item, Comp*		
Item	**Comp**	**Description**
BC4H	1,2,3,4,5,6	Second set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC5H	1,2,3,4,5,6	First set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC6H	1,2,3,4,5,6	Second set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC7H	1,2,3,4,5,6	Third set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC8H	1,2,3,4,5,6	Fourth set of six components of BCC hardness on 123 plane. Available for 3D elements only.
XELG	1,2,3,45,6,EQV	Crystal Lagrangian strain in 11, 22, 33, 12, 23,13 directions and its equivalent. Available for 3D elements only.
SINT	RHO, ETA, SSTR, GRAIN	Sintering relative density, viscosity, sintering stress, and average grain size values.

[a] Not supported by PowerGraphics.

[b] The results for this postprocessing SEND component are invalid for ELBOW290 if that element is used with viscoelastic or viscohyperelastic materials.

[c] Works only if failure criteria are provided (**FC (p. 732)** and **TB (p. 1899)**).

[d] Must first be added (**FCTYP (p. 740)**.

[e] USR1 through USR9 require a failure-criteria routine.

[f] Failure criteria are based on the effective stresses in the damaged material.

[g] For MPC-based contact definitions, the value of STAT can be negative, indicating that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.

[h] *Comp* = SUM is not supported for coupled pore-pressure-thermal (CPT*nnn*) elements.

[i] Some element- and material-type limitations apply. (See **PRERR (p. 1480)**.)

[j] Do not use **PLESOL (p. 1401)** to obtain contact force values for contact elements. (The force values reported may not be accurate for these elements.) Use **ETABLE (p. 687)** instead.

[k] WEXT is calculated for element-based loading only (and not for nodal-force loading). WEXT is stored on elements to which loading has been applied; if surface elements are added on top of other elements, for example, and pressure loading is applied to the surface elements, WEXT is available for the surface elements only.

[l] For reinforcing elements REINF264 and REINF265, issue **PLESOL (p. 1401)**,BFE,TEMP to plot the corner-point temperature of each member. You can also plot intersection-point temperature gradients (**PLESOL (p. 1401)**,TG) and intersection-point heat flux (**PLESOL (p. 1401)**,TF). For higher-order reinforcing members (generated when using higher-order base elements), the midpoint values are not available for the reinforcing members.

[m] For SOLID278 and SOLID279 with KEYOPT(3) = 2, use **PLESOL (p. 1401)**,BFE,TEMP to plot the temperature distribution through the thickness of the element. When other thermal elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are included in the model, they should be unselected to avoid plotting undefined information.

Table 213: PLESOL - Selected Result (SRES) Component Labels

PLESOL (p. 1401),SRES,*Comp*	
Comp	**Description**
SVAR*n*	The *n*th state variable.
FLDUF0*n*	The *n*th user-defined field variable.
ItemComp (concatenated *Item* + *Comp* label) from Table 208: OSRESULT - Item and Component Labels (p. 1328)[a]	See the table (p. 1328) for the combined item and component description.

[a] See Example 10: Specifying and Retrieving Selected Result Output (p. 1329).

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>Element Solu

Utility Menu>Plot>Results>Contour Plot>Elem Solution

PLETAB,*Itlab*,*Avglab*

Displays element table items.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Itlab

User-defined label, as specified with the **ETABLE (p. 687)** command, of item to be displayed.

Avglab

Averaging operation:

NOAV

Do not average element items at common nodes (default).

AVG

Average the element items at common nodes.

Notes

Displays items stored in the table defined with the **ETABLE (p. 687)** command for the selected elements. For display purposes, items are assumed to be constant over the element and assigned to each of its nodes. Contour display lines (lines of constant value) are determined by linear interpolation within each element from the nodal values. These nodal values have the option of being averaged (values are averaged at a node whenever two or more elements connect to the same node) or not averaged (discon-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tinuous). The discontinuity between contour lines of adjacent elements is an indication of the gradient across elements.

For reinforcing elements, this command displays the results of reinforcing member (individual reinforcing) selected via the **LAYER (p. 1015)**,*N* command (where *N* is a given reinforcing member). **LAYER (p. 1015)**,0 (default) or **LAYER (p. 1015)**,1 selects the first reinforcing member.

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

Menu Paths

Main Menu>General Postproc>Element Table>Plot Elem Table

Main Menu>General Postproc>Plot Results>Contour Plot>Elem Table

Utility Menu>Plot>Results>Contour Plot>Elem Table Data

PLFAR, *Lab*, *Option*, *VAR1B*, *VAR1E*, *NVAR1*, *VAR2B*, *VAR2E*, *NVAR2*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *LDSTEP*, *SUBSTEP*, *FREQB*, *FREQE*, *PlotType*, *LogOpt*

Plots pressure far fields and far-field parameters.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Parameters to plot :

PRES

Acoustic parameters

PROT

Acoustic parameters with the y-axis rotated extrusion (not valid for 2D elements)

PLAT

Acoustic parameters radiated by a vibrating structural panel (not valid for 2D elements)

Option

Plot option, based on the specified plot parameter type:

Lab	*Option*
PRES, PROT, *or* PLAT	SUMC -- Maximum pressure in Cartesian coordinate system plot (default) SUMP -- Maximum pressure in polar coordinate system plot

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab	Option
	PHSC -- Pressure phase angle in Cartesian coordinate system plot PHSP -- Pressure phase angle in polar coordinate system plot SPLC -- Sound pressure level in Cartesian coordinate system plot SPLP -- Sound pressure level in polar coordinate system plot SPAC – A-weighted sound pressure level in Cartesian coordinate system plot (dBA) SPAP – A-weighted sound pressure level in polar coordinate system plot (dBA) DGCT -- Acoustic directivity in Cartesian coordinate system plot DGPL -- Acoustic directivity in polar coordinate system plot PSCT -- Maximum scattered pressure in Cartesian coordinate system plot PSPL -- Maximum scattered pressure in polar coordinate system plot TSCT -- Target strength in Cartesian coordinate system plot TSPL -- Target strength in polar coordinate system plot PWL -- Sound power level

The arguments *VAR1B*, *VAR1E*, *NVAR1*, *VAR2B*, *VAR2E*, *NVAR2*, and *VAL1* are used when *Option* = SUMC, SUMP, PHSC, PHSP, SPLC, SPLP, SPAC, SPAP, DGCT, DGPL, PSCT, PSPL, TSCT, or TSPL:

VAR1B, VAR1E

Starting and ending values for the first variable associated with *PlotType* as described below.

When *PlotType* = blank (default) or SPHR: Starting and ending phi (ϕ) angles (in degrees) in the spherical coordinate system. Defaults to 0.

When *PlotType* = PLXY: Starting and ending x value in the Cartesian coordinate system. Defaults to 0.

When *PlotType* = PLYZ: Starting and ending y value in the Cartesian coordinate system. Defaults to 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When *PlotType* = PLXZ: Starting and ending x value in the Cartesian coordinate system. Defaults to 0.

NVAR1

Number of divisions between the starting and ending *VAR1* values for data computations. Defaults to 0.

VAR2B, VAR2E

Starting and ending values for the second variable associated with *PlotType* as described below.

When *PlotType* = blank (default) or SPHR: Starting and ending theta (θ) angles (in degrees) in the spherical coordinate system. Defaults to 0 for a 3D model and 90 for a 2D extrusion model.

When *PlotType* = PLXY: Starting and ending y value in the Cartesian coordinate system. Defaults to 0.

When *PlotType* = PLYZ: Starting and ending z value in the Cartesian coordinate system. Defaults to 0.

When *PlotType* = PLXZ: Starting and ending z value in the Cartesian coordinate system. Defaults to 0.

NVAR2

Number of divisions between the starting and ending *VAR2* values for data computations. Defaults to 0.

VAL1

VAL1 is additional input. Its meaning depends on the *PlotType* argument as described below.

When *PlotType* = blank (default) or SPHR: Radius of the sphere surface.

When *PlotType* = PLXY: Fixed z value for an X-Y plane in the Cartesian coordinate system. Defaults to 0.

When *PlotType* = PLYZ: Fixed x value for a Y-Z plane in the Cartesian coordinate system. Defaults to 0.

When *PlotType* = PLXZ: Fixed y value for an X-Z plane in the Cartesian coordinate system, Defaults to 0.

The arguments *VAL2* through *VAL5* are additional input used with certain *Lab* and *Option* settings as described below.

VAL2

When *Option* = SPLC, SPLP, SPAC, or SPAP: Reference rms sound pressure. Defaults to $2x10^{-5}$ Pa.

When *Option* = PWL: Reference sound power. Defaults to $1x10^{-12}$ watts.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL3

When *Lab* = PRES: Thickness of 2D model extrusion in the z direction (no default).

When *Lab* = PROT: Angle of the y-axis rotated extrusion model (no default).

VAL4

When *Lab* = PLAT: Mass density of acoustic fluid.

VAL5

When *Lab* = PLAT: Sound speed in acoustic fluid.

LDSTEP

Specified load step. Defaults to the load step number specified on the **SET (p. 1724)** command, or defaults to 1 if **SET (p. 1724)** has not been issued.

n

Load step number.

ALL

All load steps.

SUBSTEP

Specified substep. Defaults to the substep number specified on the **SET (p. 1724)** command, or defaults to ALL (all substeps at the specified load step) if **SET (p. 1724)** has not been issued.

n

Substep number.

ALL

All substeps.

FREQB

Frequency value representing one of the following:

- Beginning frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If a *SUBSTEP* value is specified, *FREQB* is invalid.
- Central frequency of octave bands, used when *LogOpt* = OB1, OB2, OB3, OB6, OB12, or OB24 and *FREQE* is blank.

FREQE

Ending frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If blank, *FREQE* is set to *FREQB*. If a *SUBSTEP* value is specified, *FREQE* is invalid.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PlotType

Type of plot:

ANGX

x-y chart with angle as the x-axis variable (default).

FRQX

x-y chart with frequency as the x-axis variable.

CONT

Waterfall diagram of a field parameter with variables (frequency, angle).

MRPM

Waterfall diagram of a field parameter with variables (frequency, RPM).

PLXY

Contour plot on an X-Y plane.

PLYZ

Contour plot on a Y-Z plane.

PLXZ

Contour plot on an X-Z plane.

SPHR

Contour plot on a sphere surface.

LogOpt

Octave bands (used only when Option = SPLC, SPLP, SPAC, SPAP, or PWL):

OB0

Narrow bands (default).

OB1

Octave bands.

OB2

1/2 octave bands.

OB3

1/3 octave bands.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OB6

1/6 octave bands.

OB12

1/12 octave bands.

OB24

1/24 octave bands.

Notes

The **PLFAR (p. 1412)** command plots pressure far fields and far-field parameters as determined by the equivalent source principle. Use this command to plot pressure and acoustic parameters. See the **HFSYM (p. 905)** command for the model symmetry and the **HFANG (p. 904)** command for spatial radiation angles.

Plotting acoustic parameters radiated by a vibrating structural panel (*Lab* = PLAT) is supported by elements SOLID185, SOLID186, SOLID187, SHELL181, SHELL281, and SOLSH190. The vibration surface of the panel must be flagged by the **SF (p. 1733)**,,MXWF command.

A maximum of ten curves can be plotted on a 2D chart.

The waterfall diagram is plotted only in Cartesian coordinates.

Menu Paths

This command cannot be accessed from a menu.

PLF2D,*NCONT*,*OLAY*,*ANUM*,*WIN*

Generates a contour line plot of equipotentials.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NCONT

Number of contour lines to display. Issue in multiples of 9 (that is, 9, 18, 27, etc.). Default is 27 contour lines.

OLAY

Overlay:

0

Overlay edge outlines by material number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Overlay edge outlines by real constant number.

ANUM

Highest material or real constant attribute number. Command will cycle through *ANUM* element display overlays. Defaults to 10.

WIN

Window number to which command applies. Defaults to 1.

Notes

PLF2D (p. 1417) invokes a Mechanical APDL macro which plots equipotentials of the degree of freedom AZ. The equipotential lines are parallel to flux lines and thus give a good representation of flux patterns.

In the axisymmetric case, the display is actually r * AZ where r is the node radius.

The macro overlays (*OLAY*) edge outlines by material number or real constant number (*ANUM*) and enables you to control the number of contour lines to display (*NCONT*).

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>2D Flux Lines

Utility Menu>Plot>Results>Flux Lines

PLGEOM, *Item*, *NODEkey*

Plots target and source geometries.

/MAP (p. 63): Pressure Mapping (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Items to plot:

BOTH

Plot both target and source geometries (default).

TARGET

Plot only the target geometry.

SOURCE

Plot only the source geometry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NODEkey

If the source data contains faces (that is, surface elements were created upon the **READ (p. 1578)** command), set *NODEkey* = 1 to plot only the source nodes rather than both the nodes and the elements.

Notes

Target faces are displayed in gray and source points in yellow. If the source data contains faces (that is, surface elements were created upon the **READ (p. 1578)** command), the source faces are also displayed in blue (unless *NODEkey* = 1), and both surfaces are made translucent.

Menu Paths

This command cannot be accessed from a menu.

PLLS,*LabI*,*LabJ*,*Fact*,*KUND*,*ViewUP*

Displays element table items as contoured areas along elements.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabI

Label of element table item (**ETABLE (p. 687)**) for node I magnitude.

LabJ

Label of element table item for node J magnitude.

Fact

Scale factor for display (defaults to 1). A negative scaling factor may be used to invert the display.

KUND

Undisplaced shape key:

0

Display selected items on undeformed shape.

1

Display selected items on deformed shape.

ViewUP

View Up key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Ignore the view-up (**/VUP (p. 2173)**) vector when calculating trapezoid orientation (default).

1

Use the view-up (**/VUP (p. 2173)**) vector to calculate trapezoid orientation.

Notes

Displays selected items (such as shears and moments) as a contoured area (trapezoid) display along line elements and 2D axisymmetric shell elements (such as shear and moment diagrams). Three sides of the trapezoid are formed by the element (one side) and lines at nodes I and J of length proportional to the item magnitude and displayed normal to the element and the viewing direction (the two parallel sides).

When *ViewUP* = 1, the trapezoid is oriented within the plane created by the element and the global Cartesian coordinate system reference orientation (/VUP or view up) vector. In this case, the program does not perform the calculation involving the element and view direction.

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>Line Elem Res

---

PLMAP,*Item*,--,*NODEkey*,*ImagKey*

Plots target and source pressures.

/MAP (p. 63): Pressure Mapping (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Items to plot:

BOTH

Plot both target and source pressures (default).

TARGET

Plot only the target pressures.

SOURCE

Plot only the source pressures.

--

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NODEkey

If the source data contains faces (that is, surface elements were created upon the **READ (p. 1578)** command), set *NODEkey* = 1 to plot only the source nodes rather than both the nodes and the elements.

ImagKey

0

Plot the real pressures (default).

1

Plot the imaginary pressures.

Notes

Pressures on the target faces are displayed as a color contour plot using the command **/PSF (p. 1537)**,PRES,,3. If the source data contains faces (that is, surface elements were created upon the **READ (p. 1578)** command), the source faces are also displayed using a color contour plot by default. If *NODEkey* = 1 or no source faces are available, the source pressures are displayed as colored node symbols (**/PSYMB (p. 1552)**,DOT,1 command).

Menu Paths

This command cannot be accessed from a menu.

PLMC, *LSTEP*, *SBSTEP*, *TIMFRQ*, *KIMG*, *HIbeg*, *HIend*

Plots the modal coordinates from a mode-superposition solution.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSTEP, SBSTEP

Plot the solution identified as load step *LSTEP* and substep *SBSTEP*

TIMFRQ

As an alternative to *LSTEP* and *SBSTEP*, plot the solution at the time value *TIMFRQ* (for **ANTYPE (p. 162)**,TRANS) or frequency value *TIMFRQ* (for **ANTYPE (p. 162)**,HARMIC). *LSTEP* and *SBSTEP* should be left blank.

KIMG

Key for plotting real or imaginary solution. Valid only for **ANTYPE (p. 162)**,HARMIC.

0 (or blank)

Plot the real solution (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Plot the imaginary solution.

2

Plot the amplitude.

HIbeg, HIend

For cyclic symmetry solutions, plot the solutions in the harmonic index solution range *HIbeg* to *HIend*. Defaults to all harmonic indices (all modes).

Notes

PLMC (p. 1421) plots a histogram of the modal coordinates (the factors which modes may be multiplied by to obtain their contribution to the response) at a certain time point (transient analyses) or frequency point (harmonic analyses). The absolute values of the modal coordinates are plotted. Use **/XRANGE (p. 2199)** to plot only modes in a certain range, if desired.

For transient analyses, a `.rdsp` file must be available. For harmonic analyses, a `.rfrq` file must be available. The content of these files depends on the **OUTRES (p. 1336)** command settings. Note that the default for mode-superposition transient analysis is to write the reduced displacement file every 4th substep. For more information, see Command Default (p. 1341) in the **OUTRES (p. 1336)** command description.

For a cyclic harmonic mode-superposition analysis, use the **CYCFILES (p. 457)** command to identify the `.rfrq` and modal `.rst` file. For other analyses, use the **FILE (p. 752)** command to specify the `.rdsp` or `.rfrq` file.

You may limit the plot to display only those modes in a certain harmonic index range. The modes having the same harmonic index are each plotted in a unique color. If there are less than 10 harmonic indices, they are identified in the graphics legend.

This is a graphical representation of the optional `Jobname.mcf` text file (see the **TRNOPT (p. 2017)** and **HROPT (p. 912)** commands). To print the modal coordinates, use the **PRMC (p. 1499)** command. For more information on modal coordinates, see Mode-Superposition Method in the *Mechanical APDL Theory Reference*.

Example Usage

Example Mode-Superposition Harmonic Cyclic Symmetry Analysis with Mistuning

Example: Forced Response with Mistuning and Aero Coupling

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLNEAR, *Lab*, *Opt*, *KCN*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*

Plots the pressure in the near zone exterior to the equivalent source surface.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Plot the maximum pressure or sound pressure level:

SPHERE

on the spherical structure

PATH

along the path

Opt

PSUM

Maximum complex pressure for acoustics.

PHAS

Phase angle of complex pressure for acoustics.

SPL

Sound pressure level for acoustics.

SPLA

A-weighted sound pressure level for acoustics (dBA).

KCN

KCN is the coordinate system reference number. It may be 0 (Cartesian) or any previously defined local coordinate system number (>10). Defaults to 0.

VAL1, VAL2, VAL3, . . . , VAL9

For *LAB* = SPHERE:

VAL1

Radius of spherical surface in spherical coordinate system.

VAL2

Starting ϕ angle (degree) in the spherical coordinate system. Defaults to 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL3

Ending ϕ angle (degree) in the spherical coordinate system. Defaults to 0.

VAL4

Number of divisions between the starting and ending ϕ angles for data computations. Defaults to 0.

VAL5

Starting θ angle (degrees) in the spherical coordinate system. Defaults to 0 in 3D and 90 in 2D extension.

VAL6

Ending θ angle (degrees) in the spherical coordinate system. Defaults to 0 in 3D and 90 in 2D extension.

VAL7

Number of divisions between the starting and ending θ angles for data computations. Defaults to 0.

VAL8

Reference rms sound pressure. Defaults to $2x10^{-5}$ Pa.

VAL9

Thickness of 2D model extension in z direction (defaults to 0).

For *Lab* = **PATH (p. 1354)**, **PLNEAR (p. 1423)** computes the electric field or pressure for the path data points for the path currently defined by the **PATH (p. 1354)** and **PPATH (p. 1463)** commands.

Notes

PLNEAR (p. 1423) uses the equivalent source principle to calculate the pressure in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor) for one of the following locations:

- A spherical surface in the KCN coordinate system
- A path defined by the **PATH (p. 1354)** and **PPATH (p. 1463)** commands

To plot the pressure results for a path, use the **PLPAGM (p. 1438)** or **PLPATH (p. 1439)** commands. See the **HFSYM (p. 905)** command for the model symmetry.

To retrieve saved equivalent source data, issue the **SET (p. 1724)**,*Lstep*,*Sbstep*,,REAL command.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLNSOL, *Item*, *Comp*, *KUND*, *Fact*, *FileID*, *Avg*, *DataKey*

Displays solution results as continuous element contours.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label. For selected result output, specify SRES and see Table 215: PLNSOL - Selected Result (SRES) Component Labels (p. 1434).

Comp

Component of the item (if required). Valid component labels are shown in the table below.

KUND

Undisplaced shape key:

0

Do not overlay undeformed structure display.

1

Overlay displaced contour plot with undeformed display (appearance is system-dependent).

2

Overlay displaced contour plot with undeformed edge display (appearance is system-dependent).

Fact

Scale factor for 2D display for contact items. Default value is 1. A negative scaling factor inverts the display.

FileID

The file index number (obtained via **NLDIAG (p. 1224)**,NRRE,ON). Valid only for *Item* = NRRE.

Avg

Specifies whether random acoustic results are averaged. Valid only for *Item* = U and PRES.

(blank)

No averaging (default).

AVG

Display averaged results for random acoustics.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DataKey

Key to specify which data is plotted:

AUTO

Nodal-averaged results are used, if available; otherwise, the element-based data is used, if available. (Default.)

ESOL

Only element-based results are used. If they are not available, the command is ignored.

NAR

Only nodal-averaged results are used. If they are not available, the command is ignored.

Notes

PLNSOL (p. 1425) displays the solution results as continuous contours across element boundaries for the selected nodes and elements.

For example, **PLNSOL (p. 1425)**,S,X displays the X component of stress S (that is, the SX stress component). Various element results depend upon the recalculation method and the selected results location (**AVPRIN (p. 210)**, **RSYS (p. 1639)**, **LAYER (p. 1015)**, **SHELL (p. 1784)**, and **NSEL (p. 1266)**).

Contours are determined by linear interpolation within each element from the nodal values, averaged at a node whenever two or more elements connect to the same node. (The exception is FMAG, which is summed at the node.)

For reinforcing elements (REINF*nnn*), contours are determined by interpolation within each reinforcing member of reinforcing elements from the results of the base elements. Element results of members within the same reinforcing element are smoothed based on the order of its base element. **PLNSOL (p. 1425)** displays constant results for a reinforcing element if the base elements are low-order, and linear results when the base elements are high-order.

For PowerGraphics displays (**/GRAPHICS (p. 866)**,POWER), results are plotted for the model exterior surface only. Items not supported by PowerGraphics are noted in Table 214: PLNSOL - Valid Item and Component Labels (p. 1427).

To plot midside nodes, first issue **/EFACET (p. 602)**,2.

If nodal-averaged results (**OUTRES (p. 1336)**,NAR or another nodal-averaged label) are in the database, then **PLNSOL (p. 1425)** uses the nodal-averaged data for the applicable items (S, EPEL, EPPL, EPCR, EPTH, EPSW) by default. You can change this behavior via the *DataKey* argument. Keep these points in mind when using nodal-averaged results:

- The **LAYER (p. 1015)** and **RSYS (p. 1639)**,SOLU commands are not valid with nodal-averaged results. If these commands are used, the element solution is plotted instead if applicable.
- Issuing **ESEL (p. 661)** before plotting nodal-averaged results has no effect on the output.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- PowerGraphics is supported. The output is equivalent to the full model graphics output, but only the appropriate surface nodes are plotted. See Postprocessing Nodal-Averaged Results in the *Element Reference*.)

For *Item* = SRES, selected result (**OSRESULT (p. 1327)**) values are output. See Table 215: PLNSOL - Selected Result (SRES) Component Labels (p. 1434).

Table 214: PLNSOL - Valid Item and Component Labels

General Item and Component Labels PLNSOL (p. 1425), *Item, Comp*		
Item	**Comp**	**Description**
Valid item and component labels for nodal degree of freedom results are:		
U	X, Y, Z, SUM	X, Y, or Z structural displacement or vector sum.
ROT	X, Y, Z, SUM	X, Y, or Z structural rotation or vector sum.
TEMP[a]		Temperature.
PRES		Pressure.
GFV1, GFV2, GFV3		Nonlocal field values 1, 2, and 3.
VOLT		Electric potential.
MAG		Magnetic scalar potential.
CONC		Concentration. [b]
V	X, Y, Z, SUM	X, Y, or Z fluid velocity or vector sum in a fluid analysis.
A	X, Y, Z, SUM	X, Y, or Z magnetic vector potential or vector sum in an electromagnetic analysis.
VEL	X, Y, Z, SUM	X, Y, or Z velocity or vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
ACC	X, Y, Z, SUM	X, Y, or Z acceleration or vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
OMG	X, Y, Z, SUM	X, Y, or Z rotational velocity or vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
DMG	X, Y, Z, SUM	X, Y, or Z rotational acceleration or vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
WARP		Warping.
NRRE	FX, FY, FZ, FNRM, MX, MY, MZ, MNRM	Plot the Newton-Raphson residuals from the file you obtained via the **NLDIAG (p. 1224)**,NRRE,ON command. The FNRM and MNRM labels are computed as the square root of the sum of the squares of the residual component forces or moments (FX,FY,FZ, MX, MY, MZ).[c]
		When KUND = 0, use the absolute value of the residual from the files (default).
SPL		Sound pressure level.
SPLA		A-weighted sound pressure level (dBA).
VNS		Normal velocity on the structural surface.[d]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLNSOL (p. 1425), *Item, Comp*		
Item	**Comp**	**Description**
ENKE		Acoustic diffusion energy density
Valid item and component labels for element results are:		
S	X, Y, Z, XY, YZ, XZ	Component stress.[e]
	1, 2, 3	Principal stress.[e]
	INT	Stress intensity.[e]
	EQV	Equivalent stress.[e]
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.[e]
	1, 2, 3	Principal elastic strain.[e]
	INT	Elastic strain intensity.[e]
	EQV	Elastic equivalent strain.[e]
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.[e]
	1, 2, 3	Principal thermal strain.[e]
	INT	Thermal strain intensity.[e]
	EQV	Thermal equivalent strain.[e]
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain.[b]
	1, 2, 3	Principal diffusion strain.
	INT	Diffusion strain intensity.
	EQV	Diffusion equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.[e]
	1, 2, 3	Principal plastic strain.[e]
	INT	Plastic strain intensity.[e]
	EQV	Plastic equivalent strain.[e]
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.[e]
	1, 2, 3	Principal creep strain.[e]
	INT	Creep strain intensity.[e]
	EQV	Creep equivalent strain.[e]
EPSW		Swelling strain.[e]
EPTO	X, Y, Z, XY, YZ, XZ	Component total mechanical strain (EPEL + EPPL + EPCR).
	1, 2, 3	Principal total mechanical strain.
	INT	Total mechanical strain intensity.
	EQV	Total mechanical equivalent strain.
EPTT	X, Y, Z, XY, YZ, XZ	Component total mechanical, thermal, diffusion, and swelling strain (EPEL + EPPL + EPCR + EPTH + EPDI + EPSW).
	1, 2, 3	Principal total, mechanical, thermal, diffusion, and swelling strain.
	INT	Total mechanical, thermal, diffusion, and swelling strain intensity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLNSOL (p. 1425), *Item, Comp*		
Item	**Comp**	**Description**
	EQV	Total mechanical, thermal, diffusion, and swelling equivalent strain.
ESIG	X,Y,Z,XY,YZ,ZX	Components of Biot's effective stress.
	1, 2, 3	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
	GPOR	Porosity due to void growth.
	NPOR	Porosity due to void nucleation.
NL	SEPL	Equivalent stress (from stress-strain curve).
	SRAT	Stress state ratio.
	HPRES	Hydrostatic pressure.
	EPEQ	Accumulated equivalent plastic strain.
	CREQ	Accumulated equivalent creep strain.
	PSV	Plastic state variable.
	PLWK	Plastic work/volume.
SEND	ELASTIC[f]	Elastic strain energy density. (For viscoelastic and sintering materials, the stored energy.)
	PLASTIC	Plastic strain energy density.
	CREEP	Creep strain energy density.
	DAMAGE	Damage strain energy density.
	VDAM[f]	Viscoelastic dissipation energy density.
	VREG	Visco-regularization strain energy density.
	DISS	Structural-thermal dissipation.
	ENTO	Total strain energy density (sum of ELASTIC, PLASTIC, and CREEP strain energy densities).
CDM	DMG	Damage variable.
	LM	Maximum previous strain energy for virgin material.
FAIL	MAX	Maximum of all active failure criteria defined at the current location. (See **FCTYP (p. 740)**.)[b][g]
	EMAX	Maximum Strain Failure Criterion.[b][g]
	SMAX	Maximum Stress Failure Criterion.[b][g]
	TWSI	Tsai-Wu Strength Index Failure Criterion.[b][g]
	TWSR	Inverse of Tsai-Wu Strength Ratio Index Failure Criterion.[b][g]
	HFIB	Hashin Fiber Failure Criterion.[b][g][h]
	HMAT	Hashin Matrix Failure Criterion.[b][g][h]
	PFIB	Puck Fiber Failure Criterion.[b][g][h]
	PMAT	Puck Matrix Failure Criterion.[b][g][h]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLNSOL (p. 1425), *Item, Comp*		
Item	**Comp**	**Description**
	L3FB	LaRc03 Fiber Failure Criterion.[b][g][h]
	L3MT	LaRc03 Matrix Failure Criterion.[b][g][h]
	L4FB	LaRc04 Fiber Failure Criterion.[b][g][h]
	L4MT	LaRc04 Matrix Failure Criterion.[b][g][h]
	USR1, USR2, ..., USR9	User-defined failure criteria.[b][g][h][i]
PFC	MAX[j]	Maximum of all failure criteria defined at current location.
	FT[j]	Fiber tensile failure criteria.
	FC[j]	Fiber compressive failure criteria.
	MT[j]	Matrix tensile failure criteria.
	MC[j]	Matrix compressive failure criteria.
PDMG	STAT	Damage status (0 = undamaged, 1 = damaged, 2 = completely damaged).
	FT	Fiber tensile damage variable.
	FC	Fiber compressive damage variable.
	MT	Matrix tensile damage variable.
	MC	Matrix compressive damage variable.
	S	Shear damage variable (S).
	SED	Energy dissipated per unit volume.
	SEDV	Energy per unit volume due to viscous damping.
SVAR	1, 2, 3, ... N	State variable.[b]
GKS	X, XY, XZ	Gasket component stress.
GKD	X, XY, XZ	Gasket component total closure.
GKDI	X, XY, XZ	Gasket component total inelastic closure.
GKTH	X, XY, XZ	Gasket component thermal closure.
SS	X, XY, XZ	Interface traction (stress).
SD	X, XY, XZ	Interface separation.
FICT	TEMP	Fictive temperature.
CONT[k][l]	STAT	Contact status[m]: 3 = closed and sticking 2 = closed and sliding 1 = open but near contact 0 = open and not near contact
	PENE	Contact penetration.
	PRES	Contact pressure.
	SFRIC	Contact friction stress.
	STOT	Contact total stress (pressure plus friction).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLNSOL (p. 1425), *Item, Comp*		
Item	**Comp**	**Description**
	SLIDE	Contact sliding distance.
	GAP	Contact gap distance.
	FLUX	Total heat flux at contact surface.
	CNOS	Total number of contact status changes during substep.
	FPRS	Fluid penetration pressure.
TG[n]	X, Y, Z, SUM	Component thermal gradient or vector sum.
TF[n]	X, Y, Z, SUM	Component thermal flux or vector sum.
PG	X, Y, Z, SUM	Component or vector sum of velocity or energy density flux (room acoustics).
EF	X, Y, Z, SUM	Component electric field or vector sum.
D	X, Y, Z, SUM	Component electric flux density or vector sum.
H	X, Y, Z, SUM	Component magnetic field intensity or vector sum.
B	X, Y, Z, SUM	Component magnetic flux density or vector sum.
CG	X, Y, Z, SUM	Component concentration gradient or vector sum.[b]
DF	X, Y, Z, SUM	Component diffusion flux density or vector sum.[b]
CAP	C0,X0,K0,ZONE, DPLS,VPLS	Material cap plasticity model only: Cohesion; hydrostatic compaction yielding stress; I1 at the transition point at which the shear and compaction envelopes intersect; zone = 0: elastic state, zone = 1: compaction zone, zone = 2: shear zone, zone = 3: expansion zone; effective deviatoric plastic strain; volume plastic strain.
EDPC	CSIG,CSTR	Material EDP creep model only (not including the cap model): Equivalent creep stress; equivalent creep strain.
FFLX	X,Y,Z	Fluid flow flux in poromechanics.
FGRA	X,Y,Z	Fluid pore-pressure gradient in poromechanics.
FMAG	X, Y, Z, SUM	Component electromagnetic force or vector sum.[b]
JC	X, Y, Z, SUM	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).[b]
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
PMSV	VRAT, PPRE, DSAT, RPER	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure-thermal elements.
NS	X, Y, Z, XY, YZ, XZ	Nominal strain for hyperelastic material, reported in the current configuration (unaffected by **RSYS (p. 1639)**).
MPLA	DMAC, DMAX	Microplane damage, macroscopic and maximum values.
MPDP	TOTA, TENS, COMP, RW	Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLNSOL (p. 1425), *Item, Comp*		
Item	**Comp**	**Description**
DAMAGE	1,2,3,MAX	Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).
GDMG		Damage
IDIS		Structural-thermal dissipation rate
BKS	X, Y, Z, XY, YZ, XZ	Total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements.
BKS1, . . . ,BKS5	X, Y, Z, XY, YZ, XZ	Superimposed components of the total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements when more than one superimposed back-stress component is defined.
EPFR		Free strain in porous media
SNDI	X, Y, Z, SUM	Component sound intensity or vector sum.[b]
FC1S	1,2,3,4,5,6	First set of six components of FCC crystal slip. Available for 3D elements only.
FC2S	1,2,3,4,5,6	Second set of six components of FCC crystal slip. Available for 3D elements only.
HC1S	1,2,3,4,5,6	Six components of HCP crystal slip on basal and prismatic systems. Available for 3D elements only.
HC2S	1,2,3,4,5,6	Six components of HCP crystal slip on pyramidal system. Available for 3D elements only.
HC3S	1,2,3,4,5,6	First set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC4S	1,2,3,4,5,6	Second set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC5S	1,2,3,4,5,6	Six components of HCP crystal slip on the second-order pyramidal system. Available for 3D elements only.
BC1S	1,2,3,4,5,6	First set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC2S	1,2,3,4,5,6	Second set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC3S	1,2,3,4,5,6	First set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC4S	1,2,3,4,5,6	Second set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC5S	1,2,3,4,5,6	First set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC6S	1,2,3,4,5,6	Second set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC7S	1,2,3,4,5,6	Third set of six components of BCC slip on 123 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PLNSOL (p. 1425), *Item, Comp*		
Item	**Comp**	**Description**
BC8S	1,2,3,4,5,6	Fourth set of six components of BCC slip on 123 plane. Available for 3D elements only.
FC1H	1,2,3,4,5,6	First set of six components of FCC crystal hardness. Available for 3D elements only.
FC2H	1,2,3,4,5,6	Second set of six components of FCC crystal hardness. Available for 3D elements only.
HC1H	1,2,3,4,5,6	Sixcomponents of HCP crystal hardness on basal and prismatic systems. Available for 3D elements.
HC2H	1,2,3,4,5,6	Six components of HCP crystal hardness on pyramidal system. Available for 3D elements only.
HC3H	1,2,3,4,5,6	First set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC4H	1,2,3,4,5,6	Second set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC5H	1,2,3,4,5,6	Six components of HCP crystal hardness on the second-order pyramidal system. Available for 3D elements only.
BC1H	1,2,3,4,5,6	First set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC2H	1,2,3,4,5,6	Second set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC3H	1,2,3,4,5,6	First set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC4H	1,2,3,4,5,6	Second set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC5H	1,2,3,4,5,6	First set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC6H	1,2,3,4,5,6	Second set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC7H	1,2,3,4,5,6	Third set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC8H	1,2,3,4,5,6	Fourth set of six components of BCC hardness on 123 plane. Available for 3D elements only.
XELG	1,2,3,45,6,EQV	Crystal Lagrangian strain in 11, 22, 33, 12, 23,13 directions and its equivalent. Available for 3D elements only.
SINT	RHO, ETA, SSTR, GRAIN	Sintering relative density, viscosity, sintering stress, and average grain size values.

[a] For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP to view the individual temperature degree of freedom. When other thermal elements are included in the model, deselect them to avoid plotting undefined information. To view all temperatures in the same plot, set **/ESHAPE (p. 665)**,1 and **/GRAPHICS (p. 866)**,POWER and issue **PLNSOL (p. 1425)**,TEMP.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[b] Not supported by PowerGraphics.

[c] When plotting Newton-Raphson residual items (*Item* = NRRE) from a file on the deformed geometry, the displacements are based on the current set of results in the database. These displacements may not correspond to the loadstep and substep in the .nr*xxxxx* file. (For more information about .nr*xxxxx* files and nonlinear diagnostics postprocessing, see the description of the **NLDPOST (p. 1229)** command and Performing Nonlinear Diagnostics.)

[d] Valid only for SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, and SHELL281.

[e] This item plots the solution using nodal-averaged results if they are available on the results file.

[f] The results for this postprocessing SEND component are invalid for ELBOW290 if that element is used with viscoelastic or viscohyperelastic materials.

[g] Works only if failure criteria are provided (**FC (p. 732)** and **TB (p. 1899)**).

[h] Must first be added (**FCTYP (p. 740)**).

[i] USR1 through USR9 require a failure-criteria routine.

[j] Failure criteria are based on the effective stresses in the damaged material.

[k] For contact results, PowerGraphics is supported for 3D models only.

[l] For the CONT items for elements CONTA172, CONTA174, CONTA175, and CONTA177, the reported data is averaged across the element. To obtain a more meaningful STAT value, use **PLESOL (p. 1401)**.

[m] For MPC-based contact definitions, the value of STAT can be negative, indicating that one or more contact constraints were intentionally removed to prevent overconstraint. STAT = -3 is used for MPC bonded contact; STAT = -2 is used for MPC no-separation contact.

[n] *Comp* = SUM is not supported for coupled pore-pressure-thermal (CPT*nnn*) elements.

Table 215: PLNSOL - Selected Result (SRES) Component Labels

PLNSOL (p. 1425),SRES,*Comp*	
Comp	**Description**
SVAR*n*	The *n*th state variable.
FLDUF0*n*	The *n*th user-defined field variable.
ItemComp (concatenated *Item* + *Comp* label) from Table 208: OSRESULT - Item and Component Labels (p. 1328)[a]	See the table (p. 1328) for the combined item and component description.

[a] See Example 10: Specifying and Retrieving Selected Result Output (p. 1329).

Menu Paths

Main Menu>General Postproc>Plot Results>Contour Plot>Nodal Solu

Utility Menu>Plot>Results>Contour Plot>Nodal Solution

Utility Menu>PlotCtrls>Animate>Animate Over Results

Utility Menu>PlotCtrls>Animate>Animate Over Time

Utility Menu>PlotCtrls>Animate>Deformed Results

Utility Menu>PlotCtrls>Animate>Dynamic Results

Utility Menu>PlotCtrls>Animate>Isosurfaces

Utility Menu>PlotCtrls>Animate>Mode Shape

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Utility Menu>PlotCtrls>Animate>Q-Slice Contours

Utility Menu>PlotCtrls>Animate>Time-harmonic

/PLOPTS, *Label*, *KEY*

Controls graphics options on subsequent displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Apply display items as selected from the following labels:

INFO

Controls the display of the legend (ON or OFF) and allows the choice of preset or Multi-legend placement. Control is provided by the *KEY* values. (Defaults to *KEY*=3 when the GUI is on. Defaults to *KEY*= 2 otherwise.)

LEG1

Header portion of legend column (defaults to ON).

LEG2

View portion of legend column (defaults to ON (except off with contour displays)).

LEG3

View the contour section of the legend column (defaults to ON).

FRAME

Frame border lines around windows (defaults to ON).

TITLE

Title (bottom left text) (defaults to ON).

MINM

Min-Max symbols on contour displays (defaults to ON).

LOGO

Ansys logo (defaults to OFF (displayed as text at top of legend column)). If *KEY* = ON, the text is removed from legend column but the logo symbol is displayed in whichever active window is either in the uppermost right corner and on top, or if there is no window in that location, then in the window to the furthest right of the screen. Version information remains in the legend column.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WINS

Controls whether graphics windows automatically stretch or shrink to adjust to screen size as the legend column is turned off or on (**/PLOPTS (p. 1435)**,INFO) (defaults to ON). If WINS is on and the legend column is changed from off to on, all windows are shrunk regardless of what their correct size is.

WP

Working plane (defaults to OFF). The working plane is drawn as part of the display (*not* just an overlaid image as in **WPSTYL (p. 2182)**). This option is best used in combination with a hidden-line technique (**/TYPE (p. 2034)**).

DATE

Controls the display of the date and time in your legend. Subsequent *KEY* values control the display as follows:

0 – No date or time displays are included in your legend.

1 – Only the date is shown.

2 – Both the date and time are shown (default).

FILE

Controls the display of the Mechanical APDL jobname in your legend. Subsequent *KEY* values control the display as follows:

0 – The Mechanical APDL jobname is *not* included in your legend (default).

1 – The Mechanical APDL jobname is included in your legend.

KEY

Switch:

OFF or 0

Do not apply this display item. For *Label* = DATE, no time or date are displayed.

ON or 1

Apply this display item. For *Label* = DATE, show only the date.

AUTO or 2

For *Label* = INFO, initiate Auto-legend mode. If the display has contours, the legend is ON; if the display has no contours, the legend is OFF. For *Label* = DATE, display both the date and time.

3

For *Label* = INFO , switch to Multi-legend mode. See the **/UDOC (p. 2038)** command for the available legend configurations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

See individual label defaults.

The Multi-legend mode (**/PLOPTS (p. 1435)**,INFO,3) is the default for contour legend displays.

Notes

Use **/PLOPTS (p. 1435)**,STAT to display settings. Use **/PLOPTS (p. 1435)**,DEFA to reset all specifications back to their defaults.

When you perform multiple results displays, contours on the legend column may be truncated. To avoid this, specify **/PLOPTS (p. 1435)**,LEG1,0.

The Multi-legend mode provides a number of legend data item priority and placement options. These options are accessed via the GUI at **Utility Menu> PlotCtrls> Window Controls> Window Options**. The **/UDOC (p. 2038)** command provides command line options for this capability.

This command is valid in any processor.

This command is not available for Academic Research or Teaching level products

Menu Paths

Utility Menu>PlotCtrls>Window Controls>Reset Window Options

Utility Menu>PlotCtrls>Window Controls>Window Options

PLORB

Displays the orbital motion of a rotating structure

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

When a structure is rotating and the Coriolis or gyroscopic effect is taken into account (**CORIOLIS (p. 417)**), nodes lying on the rotation axis generally exhibit an elliptical orbital motion. The **PLORB (p. 1437)** command displays the orbit of each rotating node as well as the deformed shape at time $t = 0$ (the real part of the solution).

To *print* the characteristics of the orbital path traversed by each node, issue the **PRORB (p. 1515)** command.

The **PLORB (p. 1437)** command is valid for line elements (such as BEAM188, BEAM189, PIPE288, and PIPE289). **PLORB (p. 1437)** is not supported for beam elements with the warping degree of freedom activated.

Your model must also involve a rotational velocity (**OMEGA (p. 1324)** or **CMOMEGA (p. 368)**) with Coriolis enabled (**CORIOLIS (p. 417)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Because orbit data is written in the database, a **SET (p. 1724)** command must be issued after the **PLORB (p. 1437)** command to ensure proper output for subsequent postprocessing commands.

The coordinate system for displaying nodal results must be global Cartesian (**RSYS (p. 1639)**,*KCN* = 0). **PLORB (p. 1437)** is not supported if nodes are rotated in a cylindrical coordinate system.

Menu Paths

Main Menu>General Postproc>Rotor Dynamics>Plot orbit

PLOTTING

Specifies "Plotting settings" as the subsequent status topic.

POST26 (p. 58): Status (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>TimeHist Postproc>Plot

PLPAGM, *Item*, *Gscale*, *Nopt*

Displays path items along the path geometry.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

The path data item to be displayed on the currently active path (defined by the **PATH (p. 1354)** command). Valid path items are those defined with the **PDEF (p. 1372)** or **PLNEAR (p. 1423)** commands.

Gscale

Scale factor for the offset from the path for the path data item displays. Defaults to 1.0.

Nopt

Determines how data is displayed:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(blank)

Do not display nodes, and scale the display based on the currently selected node set (default).

NODE

Display path item data along with the currently selected set of nodes. The display geometry is scaled to the selected node set.

Notes

You can use the *Gscale* argument to scale the contour display offset from the path for clarity. You need to type all six characters to issue this command.

Fore more information, see Mapping Results onto a Path in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>On Geometry

Main Menu>General Postproc>Plot Results>Plot Path Item>On Geometry

---

PLPATH, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Displays path items on a graph.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab1*, *Lab2*, *Lab3*, . . . , *Lab6

Labels identifying the path items to be displayed. Up to six items may be drawn per frame. Predefined path geometry items XG, YG, ZG, and S (**PDEF (p. 1372)**) may also be displayed.

Notes

The path must have been defined by the **PATH (p. 1354)** and **PPATH (p. 1463)** commands. Path items and their labels must have been defined with the **PDEF (p. 1372)**, **PVECT (p. 1556)**, **PCALC (p. 1362)**, **PDOT (p. 1375)**, **PCROSS (p. 1371)**, or **PLNEAR (p. 1423)** commands. Path items may also be printed with the **PRPATH (p. 1517)** command. Graph scaling may be controlled with the **/XRANGE (p. 2199)**, **/YRANGE (p. 2201)**, and **PRANGE (p. 1464)** commands. You need to type all six characters to issue this command.

Fore more information, see Mapping Results onto a Path in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>On Graph

Main Menu>General Postproc>Plot Results>Plot Path Item>On Graph

Utility Menu>Plot>Results>Path Plot

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLSECT, *Item*, *Comp*, *RHO*, *KBR*, *KBR3D*

Displays membrane and membrane-plus-bending linearized stresses.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item to be processed. Valid item labels are shown in Table 216: PLSECT - Valid Item and Component Labels (p. 1441) below. Items also require a component label.

Comp

Component of the item. Valid component labels are shown in Table 216: PLSECT - Valid Item and Component Labels (p. 1441) below.

RHO

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use a very large number (or -1) for an axisymmetric straight section.

KBR

Through-thickness bending stresses key for an axisymmetric analysis (*RHO* $\neq$ 0):

0

Include the thickness-direction bending stresses.

1

Ignore the thickness-direction bending stresses.

2

Include the thickness-direction bending stress using the same formula as the Y (axial direction) bending stress. Also use the same formula for the shear stress.

KBR3D

Through-thickness bending stresses key for 3D geometry (*RHO* = 0):

0

Include the thickness-direction bending stresses.

1

Ignore the following thickness-direction bending stresses: SX, SXY, SXZ

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Calculates and displays the membrane and membrane-plus-bending linearized stresses (as described for the **PRSECT (p. 1520)** command) along a path section (**PATH (p. 1354)**) as a graph. The path section is defined by two points specified with the **PPATH (p. 1463)** command. For linearized stress calculations, the path must be defined with nodes. The path must be entirely within the selected elements (that is, there must not be any element gaps along the path). The total stress (equivalent to the **PLPATH (p. 1439)** display) is also displayed. This command always uses 48 divisions along the path, regardless of the number of divisions defined by **PATH (p. 1354)**.

In analyses of 3D models with *RH0* = 0, ignoring the calculated out-of-plane bending stresses is recommended in some scenarios when determining the linearized bending stresses. If *KBR3D* = 0, all calculated stresses are included in the linearized bending-stress calculations. If *KBR3D* = 1, these calculated out-of-plane bending stresses are ignored in the linearized bending-stress calculations: SX, SXY, SXZ. (The principal bending-stress calculation for S1, S2, S3, SINT, and SEQV is performed with these zeroed components.)

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

Table 216: PLSECT - Valid Item and Component Labels

Item	Comp	Description
	Valid item and component labels for element results are:	
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.

Menu Paths

Main Menu>General Postproc>Path Operations>Linearized Strs

Main Menu>General Postproc>Plot Results>Plot Path Item>Lineariz Strs

---

PLTIME, *TMIN*, *TMAX*

Defines the time range for which data are to be displayed.

POST26 (p. 58): Display (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TMIN

Minimum time (defaults to the first point stored).

TMAX

Maximum time (defaults to the last point stored).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Use the previously defined range (**TIMERANGE (p. 2001)**).

Notes

Defines the time (or frequency) range (within the range stored) for which data are to be displayed. Time is always displayed in the Z-axis direction for 3D graph displays. If XVAR = 1, time is also displayed in the X-axis direction and this control also sets the abscissa scale range.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

PLTRAC,*Analopt*,*Item*,*Comp*,*TRPNum*,*Name*,*MXLOOP*,*TOLER*,*--*,*ESCL*,*MSCL*

Displays a charged particle trace on an element display.

POST1 (p. 51): Trace Points (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Analopt

Analysis option

ELEC

Particle trace in electric field

MAGN

Particle trace in magnetic field

EMAG

Particle trace in presence of both electric and magnetic fields (default)

Item

Label identifying the item to be contoured. Valid item labels are shown in Table 217: PLTRAC - Valid Item and Component Labels (p. 1444) below. Some items also require a component label. If *Item* is blank, display only the path trajectory.

Comp

Component of the item (if required). Valid component labels are shown in Table 217: PLTRAC - Valid Item and Component Labels (p. 1444) below.

TRPNum

Trace point number for storing trajectory data for use with **PATH (p. 1354)** logic. Defaults to 0 (no trajectory path data is stored for further processing with **PATH (p. 1354)** logic).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

Name of prefix of array variable. Defaults to TRAC. *Name*POIN stores trajectory path points for trace point number *TRPNum*. If *Analopt* = ELEC, MAGN, or EMAG, two additional array parameters, *Name*DATA and *Name*LABL, store trajectory path data and labels for the same *TRPNum*.

MXLOOP

Maximum number of loops traced by a particle. Defaults to 1000.

TOLER

Length tolerance used for particle trajectory geometry calculation. Valid for *Analopt* = ELEC, MAGN, or EMAG. If particle trace appears to terminate inside an element, adjusting the length tolerance may be necessary. Defaults to 1.0 x 10^{-8}.

--

Unused field.

ESCL

Electric field scale factor. Setting this scale factor affects only the tracing, not the field solution results. A negative factor corresponds to the opposite vector direction. Valid only for *Analopt* = ELEC or EMAG. Defaults to 1.

MSCL

Magnetic field scale factor. Setting this scale factor affects only the tracing, not the field solution results. A negative factor corresponds to the opposite vector direction. Valid only for *Analopt* = MAGN or EMAG. Defaults to 1.

Notes

For a specified item, the variation of the item is displayed along the particle trace as a color-contoured ribbon. The **TRPOIN (p. 2021)** command must be used to define a point on the trajectory path. Multiple traces may be displayed simultaneously by defining multiple trace points. Issue the **TRPLIS (p. 2020)** command to list the current tracing points. Issue the **TRPDEL (p. 2020)** command to delete tracing points defined earlier. Use the **PAPUT (p. 1350)** command with the POIN option to retrieve the particle trajectory points as path points.

The model must be 3D for the ELEC, MAGN, and EMAG analysis options.

Three array parameters are created at the time of the particle trace: TRACPOIN, TRACDATA and TRACLABL. These array parameters can be used to put the particle velocity and the elapsed time into path form. The procedure to put the arrays into a path named PATHNAME is as follows:

```
*get,npts,PARM,TRACPOIN,DIM,x
PATH,PATHNAME,npts,9,1
PAPUT,TRACPOIN,POINTS
PAPUT,TRACDATA,TABLES
PAPUT,TRACLABL,LABELS
PRPATH,S,T_TRACE,VX_TRACE,VY_TRACE,VZ_TRACE,VS_TRACE
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If working in the GUI, use the "All information" option to retrieve information from all three arrays at once.

Table 217: PLTRAC - Valid Item and Component Labels

Item	Comp	Description
Valid item labels for *Analopt* = ELEC nodal results are:		
VOLT		Electric potential.
Valid item labels for *Analopt* = MAGN or EMAG nodal results are:		
None		Color contour displayed.

See the *Basic Analysis Guide* for more information on charged particle traces. See Animation in the *Basic Analysis Guide* for information on particle trace animation.

Menu Paths

Main Menu>General Postproc>Plot Results>Particle Trace

Utility Menu>PlotCtrls>Animate>Particle Flow

PLVAR, *NVAR1*, *NVAR2*, *NVAR3*, *NVAR4*, *NVAR5*, *NVAR6*, *NVAR7*, *NVAR8*, *NVAR9*, *NVAR10*

Displays up to ten variables in the form of a graph.

POST26 (p. 58): Display (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR1, NVAR2, NVAR3, . . . , NVAR10

Variables to be displayed, defined either by the reference number or a unique thirty-two character name. If duplicate names are used the command will plot the data for the lowest-numbered variable with that name.

Notes

Variables are displayed vs. variable *N* on the **XVAR (p. 2199)** command. The string value will be a pre-defined, unique name. For complex variables, the amplitude is displayed by default (**PLCPLX (p. 1399)**). Each **PLVAR (p. 1444)** command produces a new frame. See the **/GRTYP (p. 874)** command for displaying multiple variables in a single frame with separate Y-axes.

Menu Paths

Main Menu>TimeHist Postpro>Graph Variables

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLVECT, *Item*, *Lab2*, *Lab3*, *LabP*, *Mode*, *Loc*, *Edge*, *KUND*

Displays results as vectors.

POST1 (p. 51): Results (p. 51)
POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Predefined vector item (from Table 218: PLVECT - Valid Item Labels (p. 1447) below) or a label identifying the i-component of a user-defined vector.

Lab2

Label identifying the j-component of a user-defined vector. In most cases, this value must be blank if *Item* is selected from Table 218: PLVECT - Valid Item Labels (p. 1447). Individual principal stresses (*Item* = S) or principal strains (*Item* = EP*xx*) may be plotted by specifying the value as 1, 2, or 3.

Lab3

Label identifying the k-component of a user-defined vector. Must be blank if *Item* is selected from list below or for 2D user defined vector.

LabP

Label assigned to resultant vector for display labeling (defaults to *Item*).

Mode

Vector or raster mode override key:

(blank)

Use the setting of *KEY* on the **/DEVICE (p. 523)** command.

RAST

Use raster mode for **PLVECT (p. 1445)** displays.

VECT

Use vector mode for **PLVECT (p. 1445)** displays.

Loc

Vector location for display of field element results:

ELEM

Display at element centroid (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NODE

Display at element nodes.

Nodal results quantities will only be displayed at nodes, not at element centroids.

Edge

Edge display override key:

(blank)

Use the setting of Key on the **/EDGE (p. 594)** command.

OFF

Deactivate the edge display.

ON

Activate the edge display.

KUND

Undisplaced shape key:

0

Display vectors on undeformed mesh or geometry.

1

Display vectors on deformed mesh or geometry.

Notes

Displays various solution results as vectors (arrows) for the selected nodes and/or elements (elements must contain at least three nodes that are not colinear). For example, **PLVECT (p. 1445)**,U displays the displacement vector for all selected nodes. For section displays (**/TYPE (p. 2034)**), the vectors are shown only on the section face (that is, cutting plane). The **PLVECT (p. 1445)** display of principal strains and stresses (*Item* = S, EPTO, EPEL, EPPL, EPCR, or EPTH) on a "cut" of the model (**/TYPE (p. 2034)**,,1 ,5,7,8, or 9) is not supported. The resulting plot displays the vectors on *all* selected elements, not on just the sliced surface. See the **/VSCALE (p. 2160)** command to scale vector lengths. Vector magnitudes may be shown as a contour display with the **PLNSOL (p. 1425)** command. Various results also depend upon the recalculation method and the selected results location (**LAYER (p. 1015)**, **SHELL (p. 1784)**, and **NSEL (p. 1266)**).

Items may be selected from a set of recognized vector labels (*Item*) or a vector may be defined from up to three scalar labels (*Item*,*Lab2*,*Lab3*). Scalar labels may be user-defined with the **ETABLE (p. 687)** command. The vectors appear on an element display as arrows showing the relative magnitude of the vector and its direction. The predefined items will be shown either at the node or at the element centroid, depending on what item is being displayed and depending on the *Loc* setting. User defined **ETABLE (p. 687)** items will be shown at the element centroid, regardless of the *Loc* setting. Stress vectors appear as arrows at the element centroid, with the arrowheads pointing away from each other for tension and toward each other for compression.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For PowerGraphics, vector arrow displays are generated in Global Cartesian (**RSYS (p. 1639)** = 0). All subsequent displays will revert to your original coordinate system.

When vector mode is active (*Mode* = VECT), use the Z-buffered display type (**/TYPE (p. 2034)**,,6) to maximize speed of **PLVECT (p. 1445)** plots (other hidden display types may make plotting slow). For PowerGraphics (**/GRAPHICS (p. 866)**,POWER), the items marked with [1] are not supported by PowerGraphics.

It is possible to plot principal stresses (*Item* = S) or principal strains (*Item* = EP*xx*) individually. To do so, specify a *Lab2* value of 1, 2, or 3. For example, the following are valid commands:

PLVECT (p. 1445),S,1,,,VECT,ELEM,ON,0

PLVECT (p. 1445),EPEL,3,,,VECT,NODE,ON,0

Table 218: PLVECT - Valid Item Labels

Item	Description
Valid item labels for nodal degree of freedom vector results are:	
U	Structural displacement vector.
ROT	Structural rotation vector.
V	Velocity vector.
A	Magnetic vector potential vector.
FFLX	Fluid flux in poromechanics.
Valid item labels for structural element results are:	
S	Principal stresses [1].
EPTO	Principal total strain (EPEL + EPPL + EPCR) [1].
EPEL	Principal elastic strains [1].
EPPL	Principal plastic strains [1].
EPCR	Principal creep strains [1].
EPTH	Principal thermal strains [1].
EPDI	Principal diffusion strains [1].
Valid item labels for field element results are:	
TG	Thermal gradient vector.
TF	Thermal flux vector.
PG	Velocity vector or energy density flux vector (room acoustics).
EF	Electric field vector.
D	Electric flux density vector.
H	Magnetic field intensity vector. If Lab2 is blank, Item is interpreted as one of the predefined labels; otherwise, Item is interpreted as a user-defined **ET (p. 686)** label and the program requests a nonblank Lab2 / Lab3 according to the dimension of the problem.
B	Magnetic flux density vector.
CG	Concentration gradient vector.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Description
DF	Diffusion flux density vector.
FMAG	Electromagnetic force vector.
P	Poynting vector.
JS	Source current density vector for low-frequency magnetic analyses. Total current density vector (sum of conduction and displacement current densities) in low frequency electric analyses.
JT	Total measureable current density vector in low-frequency electromagnetic analyses. (Conduction current density vector in a low-frequency electric analysis.)
JC	Conduction current density vector for elements that support conduction current calculation.
SNDI	Sound intensity vector [1].

1. Not supported by PowerGraphics

Menu Paths

Main Menu>General Postproc>Plot Results>Vector Plot>Predefined

Main Menu>General Postproc>Plot Results>Vector Plot>User-defined

Utility Menu>Plot>Results>Vector Plot

Utility Menu>PlotCtrls>Animate>Q-Slice Vectors

PLZZ,*RotVel*,*DeltaRotVel*

Plots the interference diagram from a cyclic modal analysis.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RotVel

Rotational speed in revolutions per minute (RPM) used to define the speed line. If blank, use the rotational speed (from **OMEGA (p. 1324)**) specified in the prestressing step of the linear perturbation analysis. If explicitly input as 0, or if the linear perturbation was not used, no speed lines are plotted.

DeltaRotVel

Adds speed lines about the *RotVel* speed line corresponding to *RotVel* ± *DeltaRotVel*. Only plotted if *RotVel* is known.

Notes

PLZZ (p. 1448) plots the cyclic modal frequencies as points on a frequency vs. harmonic index (nodal diameter) graph. If rotational speed (*RotVel*) is provided, the speed line is also plotted, leading to the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interference diagram (also known as the SAFE or ZZENF diagram). If *DeltaRotVel* is also provided, two additional speed lines are plotted, enveloping the safe speed line itself.

For more information, see Postprocessing a Modal Cyclic Symmetry Analysis in the *Cyclic Symmetry Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

/PMACRO

Specifies that macro contents be written to the session log file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command forces the contents of a macro or other input file to be written to `Jobname.log`. It is valid only within a macro or input file, and should be placed at the top of the file. **/PMACRO (p. 1449)** should be included in any macro or input file that calls GUI functions.

Menu Paths

This command cannot be accessed from a menu.

PMAP, *FORM*, *DISCON*

Creates mapping of the path geometry by defining path interpolation division points.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FORM

Defines the mapping method:

UNIFORM

Maps uniform divisions (specified on the *nDiv* argument of the **PATH (p. 1354)** command) between specified points. This is the default.

ACCURATE

Map geometry using a small division at the beginning and end of each segment. This gives you accurate derivatives, integrals, tangents, and normals for curves which do not have continuous slopes at the specified points. To create nonuniform divisions, the *nDiv* argument of the **PATH (p. 1354)** command must be greater than 2.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DISCON

Sets mapping for discontinuities in the field. The divisions are modified to put a point just before and just after the discontinuity. The valid label is MAT, for a material discontinuity. No discontinuity is the default. Discontinuity mapping involves the NOAV option on the **PDEF (p. 1372)** command.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>Path Options

Main Menu>Preprocessor>Path Operations>Define Path>Path Options

---

PMGTRAN, *Fname*, *FREQ*, *Fcnam1*, *Fcnam2*, *Pcnam1*, *Pcnam2*, *Ecnam1*, *Ccnam1*

Summarizes electromagnetic results from a transient analysis.

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name (8 characters maximum) to which tabular data and plot files will be written. Must be enclosed in single quotes when the command is manually typed in. Defaults to MG_TRNS. The data file extension is .OUT and the plot file extension is .PLT.

FREQ

Frequency of solution output. Defaults to 1. Every *FREQ*th solution on the results file is output.

Fcnam1, Fcnam2

Names of element components for force calculation. Must be enclosed in single quotes when the command is manually typed in.

Pcnam1, Pcnam2

Names of element components for power loss calculation. Must be enclosed in single quotes when the command is manually typed in.

Ecnam1, Ccnam1

Names of element components for energy and total current calculations, respectively. Must be enclosed in single quotes when the command is manually typed in.

Notes

PMGTRAN (p. 1450) invokes a Mechanical APDL macro which calculates and summarizes electromagnetic results from a transient analysis. The results are summarized by element components and listed on the screen as well as written to a file (*Fname*.OUT).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select two components for the summary of electromagnetic forces, two for power loss, and one each for stored energy (see **SENERGY (p. 1714)**) and total current (see **CURR2D (p. 444)**). See the referenced commands for other restrictions.

PMGTRAN (p. 1450) is restricted to MKSA units.

Menu Paths

Main Menu>TimeHist Postpro>Elec&Mag>Magnetics

PMLOPT,*PSYS*,*Lab*,*Xminus*,*Xplus*,*Yminus*,*Yplus*,*Zminus*,*Zplus*,*WOptXm*,*WOptXp*,*WOptYm*,*WOptYp*,*WOptZm*,*WOptZp*

Defines perfectly matched layers (PMLs) or irregular perfectly matched layers (IPML).

PREP7 (p. 22): Artificially Matched Layers (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

PSYS

PML element coordinate system number. *PSYS* may be 0 (global Cartesian) or any previously defined local Cartesian coordinate system number (>10). Defaults to 0. (Not used for IPML.)

Lab

Label defining the number of dimensions (not used for IPML):

ONE

A one-dimensional PML region.

THREE

A three-dimensional PML region (default).

Xminus

For PML, normal reflection coefficient (harmonic analysis) or attenuation factor (static structural analysis) in negative X direction of *PSYS*. Defaults to 1.E−3 (equivalent to -60 dB) for *WOptXm* = PROP or HYBR, 30 for *WOptXm* = EVAN, and 40 for *WOptXm* = MAXP.

Xplus

Normal reflection coefficient (harmonic analysis) or attenuation factor (static structural analysis) in positive X direction of *PSYS*. Defaults to 1.E−3 (equivalent to -60 dB) for *WOptXp* = PROP or HYBR, 30 for *WOptXp* = EVAN, and 40 for *WOptXp* = MAXP. (Not used for IPML.)

Yminus

Normal reflection coefficient (harmonic analysis) or attenuation factor (static structural analysis) in negative Y direction of *PSYS*. Defaults to 1.E−3 (equivalent to -60 dB) for *WOptYm* = PROP or HYBR, 30 for *WOptYm* = EVAN, and 40 for *WOptYm* = MAXP. (Not used for IPML.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Yplus

Normal reflection coefficient (harmonic analysis) or attenuation factor (static structural analysis) in positive Y direction of *PSYS*. Defaults to 1.E−3 (equivalent to -60 dB) for *WOptYp* = PROP or HYBR, 30 for *WOptYp* = EVAN, and 40 for *WOptYp* = MAXP. (Not used for IPML.)

Zminus

Normal reflection coefficient (harmonic analysis) or attenuation factor (static structural analysis) in negative Z direction of *PSYS*. Defaults to 1.E−3 (equivalent to -60 dB) for *WOptZm* = PROP or HYBR, 30 for *WOptZm* = EVAN, and 40 for *WOptZm* = MAXP. (Not used for IPML.)

Zplus

Normal reflection coefficient (harmonic analysis) or attenuation factor (static structural analysis) in positive Z direction of *PSYS*. Defaults to 1.E−3 (equivalent to -60 dB) for *WOptZp* = PROP or HYBR, 30 for *WOptZp* = EVAN, and 40 for *WOptZp* = MAXP. (Not used for IPML.)

WOptXm*, *WOptXp*, *WOptYm*, *WOptYp*, *WOptZm*, *WOptZp

Type of attenuated wave in the PML region in each direction:

PROP

Only the propagating wave is attenuated. The PML parameter is set to $s = 1–j\beta$ in harmonic analyses and acoustic transient analyses.

EVAN

Only the evanescent field is attenuated. The PML parameter is set to $s = \alpha$ ($\alpha > 1$) in static analyses.

HYBR

Both the propagating wave and the evanescent wave are attenuated (default). The PML parameter is set to $s = \alpha–j\beta$ ($\alpha > 1$). The program sets the coefficient α values in terms of the normal reflection coefficients in harmonic analyses.

MAXP

The maximum attenuation coefficient for frequency-independent PML parameter in harmonic analyses, acoustic modal analyses, and acoustic transient analyses.

Notes

The **PMLOPT (p. 1451)** command can be used to define perfectly matched layers (PML). The following element types support perfectly matched layers:

- Acoustic elements: FLUID30, FLUID220, and FLUID221 (KEYOPT(4) > 0) in a modal, harmonic, or transient acoustic analysis.
- Piezoelectric coupled-field elements: PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227 (KEYOPT(15) = 1) in a harmonic piezoelectric analysis. For the lower order elements (PLANE222 and SOLID225), PML is only supported with the B-bar method.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Structural elements: PLANE182, PLANE183, SOLID185, SOLID186, and SOLID187 (KEYOPT(15) = 1) in a linear static or harmonic structural analysis. For the lower order elements (PLANE182 and SOLID185), PML is only supported with the B-bar method.

Each PML region must have a uniquely defined PML element coordinate system (**PSYS (p. 1554)**). Normal reflection coefficient values for a harmonic analysis must be less than 1.

The **PMLOPT (p. 1451)** command can also be used to define irregular perfectly matched layers (IPML) for acoustic analyses. Normal reflection coefficient values for a harmonic analysis must be less than 1.

Issue **PMLOPT (p. 1451)**,STAT to list the current normal reflection coefficient or attenuation factor settings for a PML or IPML region. Issue **PMLOPT (p. 1451)**,CLEAR to clear all normal reflection coefficient settings and restore them to the defaults.

Issue **PMLOPT (p. 1451)**,PSYS,CLEAR to clear all normal reflection coefficient settings for the specified PML element coordinate system and restore them to the defaults.

For modal analysis, use one buffer element between the PML and the resonant structure to avoid the spurious modes. The mode patterns should be evaluated graphically.

See Artificially Matched Layers in the *Acoustic Analysis Guide* for more information about using perfectly matched layers (PML) and irregular perfectly matched layers (IPML).

Menu Paths

This command cannot be accessed from a menu.

PMLSIZE, *FREQB*, *FREQE*, *DMIN*, *DMAX*, *THICK*, *ANGLE*, *WAVESPEED*

Determines number of PML or IPML layers.

PREP7 (p. 22): Artificially Matched Layers (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FREQB

Minimum operating frequency (no default).

FREQE

Maximum operating frequency (defaults to *FREQB*).

DMIN

Minimum distance from the radiation source to the PML or IPML interface (no default).

DMAX

Maximum distance from the radiation source to the PML or IPML interface (defaults to *DMIN*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

THICK

Thickness of the PML or IPML region (defaults to 0).

ANGLE

Incident angle of wave to the PML or IPML interface (defaults to 0).

WAVESPEED

Wave speed in PML or IPML medium (defaults to 343.24 m/s).

Notes

PMLSIZE (p. 1453) determines the number of PML or IPML layers for acceptable numerical accuracy.

PMLSIZE (p. 1453) must be issued before any meshing commands. If the thickness of the PML or IPML region is known, it determines an element edge length (h) and issues **ESIZE (p. 668)**,h. If the thickness of the PML or IPML region is unknown, it determines the number of layers (n) and issues **ESIZE (p. 668)**,,n.

See Artificially Matched Layers in the *Acoustic Analysis Guide* for more information about using perfectly matched layers (PML) and irregular perfectly matched layers (IPML).

Menu Paths

This command cannot be accessed from a menu.

/PMORE, - -, *X5*, *Y5*, *X6*, *Y6*, *X7*, *Y7*, *X8*, *Y8*

Creates an annotation polygon (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

- -

Unused field.

X5

X location for vertex 5 of polygon (-1.0 < X < 2.0).

Y5

Y location for vertex 5 of polygon (-1.0 < Y < 1.0).

X6

X location for vertex 6 of polygon (-1.0 < X < 2.0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y6

Y location for vertex 6 of polygon (-1.0 < Y < 1.0).

X7

X location for vertex 7 of polygon (-1.0 < X < 2.0).

Y7

Y location for vertex 7 of polygon (-1.0 < Y < 1.0).

X8

X location for vertex 8 of polygon (-1.0 < X < 2.0).

Y8

Y location for vertex 8 of polygon (-1.0 < Y < 1.0).

Notes

Defines the 5th through 8th vertices of an annotation polygon (**/POLYGON (p. 1460)**). This is a command generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if annotation is used.

The command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with the **/INPUT (p. 948)** command).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

PNGR,*Keyword*,*OPT*,*VAL*

Provides PNG file export for Mechanical APDL displays.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Keyword

Specifies various PNG file export options.

COMP

If *Keyword* = COMP, then *OPT* is either ON or OFF (blank is interpreted as OFF). This option allows you to turn PNG file compression ON or OFF. If OPT = ON, then The VAL field is read to determine the degree of compression. See the VALUE argument for acceptable compression values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ORIENT

If *Keyword* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

COLOR

If *Keyword* = COLOR, then *OPT* will determine the color depth of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

TMOD

If *Keyword* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

DEFAULT

If *Keyword* = DEFAULT, then all of the default values, for all of the Keyword parameters listed above, are active.

STAT

Shows the current status of PNG file export.

OPT

OPT can have the following names or values, depending on the value for *Keyword* (see above).

ON, OFF

If *Keyword* = COMP, the values On and Off control the use of compression. The degree of compression is determined by VAL

Horizontal, Vertical

If *Keyword* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

0, 1, 2

If *Keyword* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

1, 0

If *Keyword* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

VAL

VAL is active only when *Keyword* = COMP, and determines the degree of compression applied to the exported file (see above).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

-1

Apply the default, optimum value for compression. This value represents the best combination of speed and compression. It varies according to the release level of the ZLIB compression package.

1-9

Use this value to specify a specific compression level. 1 is the lowest compression level (fastest) and 9 is the highest compression level (slowest).

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots

Utility Menu>PlotCtrls>Hard Copy

/PNUM, *Label*, *KEY*

Controls entity numbering/coloring on plots.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Type of numbering/coloring:

NODE

Node numbers on node and element plots.

ELEM

Element numbers and colors on element plots.

SEC

Section numbers and colors on element and solid model plots (see "Notes" (p. 1459)).

MAT

Material set numbers and colors on element and solid model plots (see "Notes" (p. 1459)).

TYPE

Element type reference numbers and colors on element and solid model plots (see "Notes" (p. 1459)).

REAL

Real constant set numbers and colors on element and solid model plots (see "Notes" (p. 1459)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ESYS

Element coordinate system numbers on element and solid model plots (see "Notes" (p. 1459)).

LOC

Location numbers/colors of the element in matrix assembly order on element plots.

Note:

LOC and ELEM numbers will be the same unless the model has been reordered.

KP

Keypoint numbers on solid model plots.

LINE

Line numbers on solid model plots (both numbers and colors on line plots).

AREA

Area numbers on solid model plots (both numbers and colors on area plots).

VOLU

Volume numbers on solid model plots (both numbers and colors on volume plots).

SVAL

Stress (or any contour) values on postprocessing plots, and surface load values and colors on model plots when surface load symbols are on (**/PSF (p. 1537)**). For tabular boundary conditions, the table-evaluated values will be displayed on node, element, or contour displays in POST1 when load symbols (**/PBF (p. 1360)**, **/PSF (p. 1537)**, **/PBC (p. 1356)**) are on and TABNAM is OFF.

TABNAM

Table names for tabular boundary conditions. If this label is turned on, the table name appears next to the appropriate symbol, arrow, face outline, or contour as dictated by the **/PSF (p. 1537)**, **/PBC (p. 1356)**, and **/PBF (p. 1360)** commands.

STAT

Shows current settings for **/PNUM (p. 1457)**.

DEFA

Resets all **/PNUM (p. 1457)** specifications back to default.

KEY

Switch:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Turns OFF numbers/colors for specified label.

1

Turns ON numbers/colors for specified label.

Notes

This command specifies entity numbering and coloring for subsequent plots.

The MAT, TYPE, REAL, and ESYS labels activate both the numbering and coloring of the corresponding attributes for **EPLOT (p. 648)**, **KPLOT (p. 988)**, **LPLOT (p. 1061)**, **APLOT (p. 169)**, and **VPLOT (p. 2147)**. The ELEM, MAT, TYPE, REAL, ESYS, and LOC labels are mutually exclusive, that is, only one can be specified at a time. Also, turning on a LINE, AREA, or VOLU label will turn off the MAT, TYPE, and REAL labels.

PowerGraphics (**/GRAPHICS (p. 866)**,POWER) displays for**/PNUM (p. 1457)** can be problematic. **/PNUM (p. 1457)**,ELEM will display erratically depending on other display command specifications, while **/PNUM (p. 1457)**,LOC and **/PNUM (p. 1457)**,ESYS are not supported.

Element and volume numbers are not visible for 3D elements and volumes when Z-buffering is turned on (**/TYPE (p. 2034)**,,[6,7, or 8]).

Use **/PSTATUS (p. 1550)** or **/PNUM (p. 1457)**,STAT to show settings. Use **/PNUM (p. 1457)**,DEFA to reset all specifications back to default. Use the **/NUMBER (p. 1290)** command to control whether numbers and colors are displayed together.

This command is valid in any processor

Menu Paths

Utility Menu>PlotCtrls>Numbering

POINT

Specifies "Point flow tracing settings" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>List>Status>General Postproc>Trace Points

POLY

Creates a polygonal area based on working plane coordinate pairs.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Defines a polygonal area on the working plane. The area will be defined with NPT keypoints and NPT lines, where NPT (must be at least 3) is the number of coordinate pairs defined with the **PTXY (p. 1556)** command. See the **RPOLY (p. 1616)** and **RPR4 (p. 1617)** commands for other ways to create polygons.

Menu Paths

This command cannot be accessed from a menu.

/POLYGON, *NVERT*, *X1*, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*

Creates annotation polygons (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVERT

Number of vertices of polygon (3 ≤ *NVERT* ≤ 8). Use **/PMORE (p. 1454)** for polygons with more than 4 vertices.

X1

X location for vertex 1 of polygon (-1.0 < X < 2.0).

Y1

Y location for vertex 1 of polygon (-1.0 < Y < 1.0).

X2

X location for vertex 2 of polygon (-1.0 < X < 2.0).

Y2

Y location for vertex 2 of polygon (-1.0 < Y < 1.0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X3

X location for vertex 3 of polygon (-1.0 < X < 2.0).

Y3

Y location for vertex 3 of polygon (-1.0 < Y < 1.0).

X4

X location for vertex 4 of polygon (-1.0 < X < 2.0).

Y4

Y location for vertex 4 of polygon (-1.0 < Y < 1.0).

Notes

Creates annotation polygons to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.log`) if annotation is used.

The command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

All polygons are shown on subsequent displays unless the annotation is turned off or deleted. Issue **/LSPEC (p. 1081)** and **/PSPEC (p. 1548)** to set the attributes of the polygon. Issue **/PMORE (p. 1454)** to define the 5th through 8th vertices of the polygon.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

/POST1

Enters the database results postprocessor.

SESSION (p. 11): Processor Entry (p. 12)
POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the general database results postprocessor (POST1). All load symbols (**/PBC (p. 1356)**, **/PSF (p. 1537)**, or **/PBF (p. 1360)**) are automatically turned off with this command.

This command is valid only at the Begin Level.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc

/POST26

Enters the time-history results postprocessor.

SESSION (p. 11): Processor Entry (p. 12)
POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the time-history results postprocessor (POST26).

This command is valid only at the Begin Level.

Menu Paths

Main Menu>TimeHist Postpro

POWERH

Calculates the rms power loss in a conductor or lossy dielectric.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

POWERH (p. 1462) invokes a Mechanical APDL macro which calculates the time-averaged (rms) power loss in a conductor or lossy dielectric material from a harmonic analysis. The power loss is stored in the parameter PAVG.

Conductor losses include solid conductors and surface conductors approximated by impedance or shielding boundary conditions. The power-loss density for solid conductors or dielectrics is stored in the element table with the label PLOSSD and may be listed (**PRETAB (p. 1490)**) or displayed (**PLETAB (p. 1411)**). PLOSSD does not include surface losses.

The elements of the conducting region must be selected before this command is issued.

POWERH (p. 1462) is valid for 2D and 3D analyses.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Power Loss

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PPATH, *POINT*, *NODE*, *X*, *Y*, *Z*, *CS*

Defines a path by picking or defining nodes, or locations on the currently active working plane, or by entering specific coordinate locations.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

POINT

The point number. It must be greater than zero and less than or equal to the *nPts* value specified on the **PATH (p. 1354)** command if graphical picking is not being used.

NODE

The node number defining this point. If blank, use the X, Y, Z coordinates to define the point. A valid node number will override *X*, *Y*, *Z* coordinate arguments.

X, Y, Z

The location of the point in the global Cartesian coordinate system. Use these arguments only if you omit the *NODE* argument.

CS

The coordinate system for interpolation of the path between the previous point and this point. Omit this argument if you wish to use the currently active (**CSYS (p. 441)**) coordinate system. If the coordinate system of two adjacent points is different, the *CS* value of the latter point will be used.

Notes

For linearized stress calculations, the path must be defined with nodes.

This command is designed and works best in interactive (GUI) mode, using the menu paths listed below. For command line operations, issue **PPATH (p. 1463)**,P to define your path by picking nodes.

For information on displaying paths you have defined, see Mapping Results onto a Path in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>By Location

Main Menu>General Postproc>Path Operations>Define Path>By Nodes

Main Menu>General Postproc>Path Operations>Define Path>Modify Path

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Define Path>By Location

Main Menu>Preprocessor>Path Operations>Define Path>By Nodes

Main Menu>Preprocessor>Path Operations>Define Path>Modify Path

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRANGE, *LINC*, *VMIN*, *VMAX*, *XVAR*

Determines the path range.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINC*, *VMIN*, *VMAX

Set the range for listing or displaying the table locations between a minimum value (*VMIN*) and a maximum value (*VMAX*) of the path distance with a location increment of *LINC* (defaults to 1). The first location begins at *VMIN*.

XVAR

Path variable item to be used as the x-axis plot variable. Any valid path variable may be used (**PDEF (p. 1372)** command). Default variable is the path distance, S.

Command Default

Include every interpolation point and entire path distance.

Notes

Determines the path distance range for use with the **PRPATH (p. 1517)** and **PLPATH (p. 1439)** commands.

Menu Paths

Main Menu>General Postproc>Path Operations>Plot Path Item>Path Range

PRAS, *Lab*, *LDSTEP*, *SUBSTEP*, *FREQB*, *FREQE*, *LogOpt*, - -, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*

Prints a specified acoustic quantity during postprocessing of an acoustic analysis.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

The acoustic quantity to calculate:

SIMP

Specific acoustic impedance on the selected surface.

AIMP

Acoustic impedance on the selected surface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MIMP

Mechanical impedance on the selected surface.

PRES

Average pressure on the selected surface.

FORC

Force on the selected surface.

POWE

Acoustic power on the selected surface.

ERP

Equivalent radiated power on the selected structural surface (valid only for SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, SOLID225, SOLID226, SOLID227, and SHELL281).

ERPL

Equivalent radiated power level on the selected structural surface (valid only for SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, SOLID225, SOLID226, SOLID227, and SHELL281).

BSPL

Frequency-band sound pressure level on selected nodes.

BSPA

A-weighted frequency-band sound pressure level on selected nodes.

MENE

Acoustic potential energy on the selected elements.

KENE

Acoustic kinetic energy on the selected elements.

TENE

Acoustic total energy on the selected elements.

PL2V

Average square of the L2 norm of pressure on the selected elements.

LWIN

Input sound power level on defined port.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LWOUT

Output sound power level on defined driven port.

RL

Return loss on defined port.

ALPHA

Absorption coefficient on defined port.

TL

Transmission loss on defined ports.

PALL

All port-related parameters (LWIN, LWOUT, RL, ALPHA, TL).

DFSTL

Transmission loss of random acoustic analysis.

DFSPW

Radiated power in random acoustic analysis.

DALL

All random acoustic related parameters (DFSTL, DFSPW).

LDSTEP

Specified load step. Defaults to the load step number specified on the **SET (p. 1724)** command, or defaults to 1 if **SET (p. 1724)** has not been issued. This default applies to all *Lab* values except DFSTL, DFSPW, and DALL.

n

Load step number.

ALL

All load steps.

AVG or 0

Average result of multiple samplings in a random acoustic analysis (see the **MSOLVE (p. 1193)** command). This option is used only for *Lab* = DFSTL, DFSPW, and DALL, and it is the default for these labels.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SUBSTEP

Specified substep. Defaults to the substep number specified on the **SET (p. 1724)** command, or defaults to ALL (all substeps at the specified load step) if **SET (p. 1724)** has not been issued. For *Lab* = BSPL or BSPA, ALL is the only valid value.

n

Substep number.

ALL

All substeps.

FREQB

Frequency value representing one of the following:

- Beginning frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If a *SUBSTEP* value is specified, *FREQB* is invalid.
- Central frequency of octave bands, used when *LogOpt* = OB1, OB2, OB3, OB6, OB12, or OB24 and *FREQE* is blank.

FREQE

Ending frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If blank, *FREQE* is set to *FREQB*. If a *SUBSTEP* value is specified, *FREQE* is invalid.

LogOpt

Octave bands:

OB0

Narrow bands (default).

OB1

Octave bands.

OB2

1/2 octave bands.

OB3

1/3 octave bands.

OB6

1/6 octave bands.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OB12

1/12 octave bands.

OB24

1/24 octave bands.

- -

Unused field.

VAL1

Input port number for *Lab* = LWIN, LWOUT, RL, ALPHA, TL, or PALL.

VAL2

Output port number for *Lab* = TL or PALL.

VAL3

Reference power for *Lab* = LWIN, LWOUT, PALL or EPRL (defaults to $1x10^{-12}$ W).

VAL4

Fluid mass density for *Lab* = ERP or ERPL (defaults to 1.2041 kg/m^3).

VAL5

Speed of sound in the fluid for *Lab* = ERP or ERPL (defaults to 343.25 m/s).

VAL6

Radiation efficiency for *Lab* = ERP or ERPL (defaults to 1).

Notes

The **PRAS (p. 1464)** command lists the specified acoustic quantity on the selected exterior surface, the energy on selected elements, or the sound pressure level over frequency bands. The calculation is based on the pressure and velocity solution or the frequency-band sound pressure level (SPL).

The total pressure and velocity are used if the selected surface is the excitation source surface. To calculate the incoming and outgoing acoustic power and other sound power parameters on the input and output surfaces, issue the **SF (p. 1733)**,,PORT command in the preprocessor to define port numbers.

The sound pressure level of the octave bands and general frequency band (defined via the **HARFRQ (p. 885)** command) is calculated at the selected nodes in the model.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRCAMP,*Option*,*SLOPE*,*UNIT*,*FREQB*,*Cname*,*STABVAL*,*KeyALLFreq*,*KeyNegFreq*,*KeyWhirl*

Prints Campbell diagram data for applications involving rotating structure dynamics.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Flag to activate or deactivate sorting of forward or backward whirl frequencies:

0 (OFF *or* NO)

No sorting.

1 (ON *or* YES)

Sort. This value is the default.

SLOPE

The slope of the line to be printed. This value must be positive.

SLOPE > 0

In the stationary reference frame (*RefFrame* = YES on the **CORIOLIS (p. 417)** command), the line represents the number of excitations per revolution of the rotor. For example, *SLOPE* = 1 represents one excitation per revolution, usually resulting from unbalance.

In the rotating reference frame (*RefFrame* = NO on the **CORIOLIS (p. 417)** command), the line represents the number of excitations per revolution of the rotor minus 1.

SLOPE = 0

The line represents the stability threshold for stability values or logarithmic decrements printout (*STABVAL* = 1, 2, or 3)

UNIT

Specifies the unit of measurement for rotational angular velocities:

RDS

Rotational angular velocities in radians per second (rad/s). This value is the default.

RPM

Rotational angular velocities in revolutions per minute (RPMs).

FREQB

The beginning, or lower end, of the frequency range of interest. The default is zero.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cname

The rotating component name.

STABVAL

Flag to print the stability values:

0 (OFF *or* NO)

Print the frequencies (the imaginary parts of the eigenvalues in Hz). This value is the default.

1 (ON *or* YES)

Print the stability values (the real parts of the eigenvalues in Hz).

2

Print the inverse of the logarithmic decrements. A negative logarithmic decrement indicates stable motion.

3

Print the logarithmic decrements. A positive logarithmic decrement indicates stable motion and is consistent with API (American Petroleum Institute) standards.

For more information about complex eigenmodes and corresponding logarithmic decrements, see Complex Eigensolutions in the *Mechanical APDL Theory Reference*.

KeyALLFreq

Key to specify if all frequencies above FREQB are printed out:

0 (OFF *or* NO)

A maximum of 10 frequencies are printed out. They correspond to the frequencies displayed via the **PLCAMP (p. 1392)** command. This value is the default.

1 (ON *or* YES)

All frequencies are printed out.

KeyNegFreq

Key to specify if the negative frequencies are printed out. It only applies to solutions obtained with the damped eigensolver (*Method*=DAMP on the **MODOPT (p. 1140)** command):

0 (OFF *or* NO)

Only positive frequencies are printed out. This value is the default.

1 (ON *or* YES)

Negative and positive frequencies are printed out.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KeyWhirl

Flag to print the whirl and instability keys for each load step:

0 (OFF *or* NO)

Print the whirl for the last load step. This value is the default.

1 (ON *or* YES)

Print the whirl and instability keys for each load step.

Notes

The following items are required when generating a Campbell diagram:

- Activate the Coriolis effect (**CORIOLIS (p. 417)** command) in the solution phase (**/SOLU (p. 1821)**).
- Run a modal analysis using the QR damped (**MODOPT (p. 1140)**,QRDAMP) or damped (**MODOPT (p. 1140)**,DAMP) method. Complex eigenmodes are necessary (**MODOPT (p. 1140)**,QRDAMP,,,,*Cpxmod* = ON), and you must specify the number of modes to expand (**MXPAND (p. 1203)**).
- Define two or more load step results with an ascending order of rotational velocity (**OMEGA (p. 1324)** or **CMOMEGA (p. 368)**).

In some cases where modes are not in the same order from one load step to the other, sorting the frequencies (*Option* = 1) can help to obtain a correct printout. Sorting is based on the comparison between complex mode shapes calculated at two successive load steps.

At each load step, the application compares the mode shape to the loads to determine the whirl direction. If applicable, a label appears (on the rows of output data) representing the whirl mode (BW for backward whirl and FW for forward whirl).

If you specify a non-zero slope (*SLOPE* > 0), the command prints the critical speeds corresponding to the intersection points of the frequency curves and the added line. In the case of a named component (*Cname*), critical speeds relate to the rotational velocity of the component. Critical speeds are available only if the frequencies are printed (*STABVAL* = OFF).

If you specify a zero slope (*SLOPE* = 0), the command prints the stability threshold corresponding to the sign change of the stability values (or logarithmic decrements). In the case of a named component (*Cname*), stability thresholds relate to the rotational velocity of the component. Stability thresholds are available only if the stability values or logarithmic decrements are printed (*STABVAL* = 1, 2, or 3).

At each load step, the program checks for instability (based on the sign of the real part of the eigenvalue and a damping ratio with absolute value greater than 0.1%). The label "U" appears on the printout for each unstable frequency.

If specified, the rotational velocities of the named component (*Cname*) are printed out along with the natural frequencies.

For information on printing a Campbell diagram for a prestressed structure, see Solving for a Subsequent Campbell Analysis of a Prestressed Structure Using the Linear Perturbation Procedure in the *Rotordynamic Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a usage example of the companion command **PLCAMP (p. 1392)** (used for *plotting* a Campbell diagram), see Example: Campbell Diagram Analysis of a Simply Supported Beam in the *Rotordynamic Analysis Guide*.

For more information on Campbell diagram generation, see Campbell Diagram in the *Rotordynamic Analysis Guide*.

Damped modal cyclic symmetry (**CYCLIC (p. 462)**) analyses do not support the **PRCAMP (p. 1469)** command.

Menu Paths

Main Menu>General Postproc>Rotor Dynamics>Print Campbell

PRCINT, *ID*, *Node*, *Dtype*

Lists fracture parameter (CINT (p. 334)) results data.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ID

Crack ID number.

Node

Crack tip node number. Default = ALL. Valid only for 3D analysis.

Dtype

Data type to output:

JINT

J-integral

IIN1

Interaction integral 1

IIN2

Interaction integral 2

IIN3

Interaction integral 3

K1

Mode 1 stress-intensity factor

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

K2

Mode 2 stress-intensity factor

K3

Mode 3 stress-intensity factor

G1

Mode 1 energy release rate

G2

Mode 2 energy release rate

G3

Mode 3 energy release rate

GT

Total energy release rate

MFTX

Total material force X

MFTY

Total material force Y

MFTZ

Total material force Z

TSTRESS

T-stress

CEXT

Crack extension

CSTAR

C*-integral

STTMAX

Maximum circumferential stress

PSMAX

Maximum circumferential stress when $\sigma_{r\theta}$=0

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DLTA

Incremental crack extension in a fatigue/static crack-growth analysis

DLTN

Number of incremental cycles in a fatigue/static crack-growth analysis

DLTK

Equivalent stress-intensity factor range in a fatigue/static crack-growth analysis

KEQV

Equivalent stress-intensity factor in a fatigue/static crack-growth analysis

KANG

Kink angle (degree) in a fatigue/static crack-growth analysis

R

Stress (load) ratio in a fatigue/static crack-growth analysis

UFAC

U-factor (crack closure) in a fatigue/static crack-growth analysis

CRDX

X coordinate of the crack tip

CRDY

Y coordinate of the crack tip

CRDZ

Z coordinate of the crack tip

APOS

Position attribute of the crack-tip node:

Positive integer –

The subcrack[1] ID number to which this tip belongs. For a crack with only a single subcrack, this value is 1.

Negative integer –

The absolute value of the negative integer is the subcrack ID number to which this tip belongs.

[1] Subcracks typically appear in SMART crack-growth analyses and are uncommon in other types of fracture analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The negative sign indicates that this crack tip is the end of this subcrack, and that this subcrack is a closed polygon. It must be connected to the first tip of this subcrack when the crack front is plotted.

For more information, see "Examples: APOS Usage" (p. 1475).

Examples: APOS Usage

The following examples show how APOS values[2] are applied in several cases for fracture analysis.

The most common situation is that an open crack exists in *N* crack tips, and all tips are connected into a single subcrack. The APOS values for each tip are:

Tip Index	1	2	3	… …	N-1	N
APOS	1	1	1	1… …1	1	1

For a closed crack without extra subcracks, the APOS values are:

Tip Index	1	2	3	… …	N-1	N[a]
APOS	1	1	1	1… …1	1	**-1**

[a] The value for the N^{th} tip is -1, meaning that the crack front should be plotted as a closed loop.

The following crack has two subcracks, the first open and the second closed. Assuming *M* tips on the first subcrack and *N* tips on the second, the APOS values are:

Tip Index	1	… …	M	M+1	… …	M+N
APOS	1	1	1	2	2	**-2**

Notes

When a crack tip node is defined, the values associated with the specified node are listed.

Dtype = STTMAX or PSMAX are valid for phantom-node-based XFEM analyses only.

Dtype = CRDX, CRDY, CRDZ, and APOS are valid only in a fatigue/static crack-growth analysis using SMART or singularity-based XFEM.

Dtype = DLTA, DLTN, DLTK, R are valid in a fatigue crack-growth analysis using SMART or singularity-based XFEM. DLTA is also valid for static crack-growth analysis using SMART.

Dtype = UFAC is valid only in a fatigue crack-growth analysis using SMART.

Dtype = KEQV and KANG are valid only in a fatigue/static crack-growth analysis using SMART.

[2] Issuing ***GET (p. 797)** is an effective way to obtain APOS values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

PRCPLX, *KEY*

Defines the output form for complex variables.

POST26 (p. 58): Listing (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Output form key:

0

Real and imaginary parts.

1

Amplitude and phase angle. Stored real and imaginary data are converted to amplitude and phase angle upon output. Data remain stored as real and imaginary parts.

Notes

Defines the output form for complex variables. Used only with harmonic analyses (**ANTYPE (p. 162)**,HARMIC).

All results data are stored in the form of real and imaginary components and converted to amplitude and/or phase angle as specified via the **PRCPLX (p. 1476)** command. The conversion is not valid for derived results (such as principal stress/strain, equivalent stress/strain and USUM).

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

PRED, *Sskey*, - -, *Lskey*

Activates a predictor in a nonlinear analysis.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sskey

Substep predictor key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AUTO

The program uses a predictor but, within certain exceptions, automatically switches prediction off. This behavior is the default; see "Command Default" (p. 1477) for details.

OFF

No prediction occurs.

LINEAR (or ON)

Use the linear predictor on all substeps after the first.

QUADRATIC

Use the quadratic predictor on all substeps after the second.

- -

Unused field.

Lskey

Load step predictor:

OFF

No prediction across load steps occurs. This is the default behavior.

ON

Use a predictor also on the first substep of the load step. (*Sskey* = ON is required.)

Command Default

The default command behavior is to use prediction (*Sskey* = AUTO). The AUTO option chooses to either use the linear predictor or to turn the predictor OFF. However, prediction does not occur if one or more of these conditions exist:

- Over prediction occurs due to a large residual force or excessive element distortion.
- You are mapping (**MAPSOLVE (p. 1108)**) variables to a new mesh during rezoning. (Prediction does not occur for any **MAPSOLVE (p. 1108)** substeps, nor for the first substep afterwards.)
- You have steady-state analysis defined (**SSTATE (p. 1855)**), and contact elements exist in the model.

Notes

Activates a predictor in a nonlinear analysis on the degree-of-freedom solution for the first equilibrium iteration of each substep.

When using the arc-length method (**ARCLEN (p. 175)**, **ARCTRM (p. 177)**), you cannot issue the DOF solution predictor command (**PRED (p. 1476)**), the automatic time stepping command (**AUTOTS (p. 207)**), or the line search command (**LNSRCH (p. 1057)**). If you activate the arc-length method *after* you set

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRED (p. 1476), **AUTOTS (p. 207)**, or **LNSRCH (p. 1057)**, a warning message appears. If you elect to proceed with the arc-length method, the program disables your DOF predictor, automatic time stepping, and line search settings, and the time step size is controlled by the arc-length method internally.

When using step-applied loads, such as **TUNIF (p. 2027)**, **BFUNIF (p. 259)**, etc., or other types of non-monotonic loads, the predictor may adversely affect the convergence. If the solution is discontinuous, the predictor may need to be turned off.

When performing a nonlinear analysis involving large rotations, the predictor may require using smaller substeps. If the model has rotational degrees-of-freedom, the quadratic predictor could work more efficiently than the linear predictor.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Predictor

Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Solution>Load Step Opts>Nonlinear>Predictor

PRENERGY,*EnergyType*,*Cname1*,*Cname2*,*Cname3*,*Cname4*,*Cname5*,*Cname6*

Prints the total energies of a model or the energies of the specified components.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

EnergyType

Type of energies to be printed:

ALL

All energies are printed: potential, kinetic, artificial hourglass/drill stiffness, contact stabilization energy, and artificial stabilization energy when applicable. This is the default.

SENE

Potential energy (stiffness energy).

KENE

Kinetic energy.

DENE

Damping energy.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WEXT

Work done by external loads.

Cname1, Cname2, Cname3,...

Component names for energies of the components printout.

If *Cname1* is blank, the total energies are listed.

If *Cname1* = ALL, the energies are listed for all selected components.

If *Cname1* is neither blank nor ALL, it is the name of an existing component. The energies are listed for up to 6 selected components named in *Cname1* to *Cname6*.

Notes

The **PRENERGY (p. 1478)** command prints out either the total energies of the entire model or the energies of the components depending on the *Cname1* specification.

Only existing components based on elements (defined with the **CM (p. 356)** command) are supported when component energies are listed.

Damping energy (DENE) and work done by external loads (WEXT) are available only if the following were set prior to the analysis solution: *EngCalc* = YES on the **TRNOPT (p. 2017)**, **HROUT (p. 918)** or **MXPAND (p. 1203)** command; and *Item* = VENG, ESOL, or ALL on the **OUTRES (p. 1336)** command.

If *EngCalc* = YES on the **HROUT (p. 918)** or **MXPAND (p. 1203)** command, average, amplitude, and peak values are returned for potential (SENE) and kinetic (KENE) energies.

The energy values can be retrieved using the ***GET (p. 797)** command with *Entity* = PRENERGY.

This command applies to structural elements only.

Menu Paths

This command cannot be accessed from a menu.

/PREP7

Enters the model creation preprocessor.

SESSION (p. 11): Processor Entry (p. 12)
PREP7 (p. 22): Database (p. 22)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Enters the general input data preprocessor (PREP7).

This command is valid only at the Begin Level.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor

PRERR

Prints SEPC and TEPC.

POST1 (p. 51): Listing (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Prints the percent error in structural energy norm (SEPC) and the thermal energy norm percent error (TEPC). Approximations of mesh discretization error associated with a solution are calculated for analyses having structural or thermal degrees of freedom.

The structural approximation is based on the energy error (which is similar in concept to the strain energy) and represents the error associated with the discrepancy between the calculated stress field and the globally continuous stress field (see POST1 - Error Approximation Technique in the *Mechanical APDL Theory Reference*). This discrepancy is due to the assumption in the elements that only the displacements are continuous at the nodes. The stress field is calculated from the displacements and should also be continuous, but generally is not.

Thermal analyses may use any solid and shell thermal element having only temperature degrees of freedom. The thermal approximation is based on the total heat flow dissipation and represents the error associated with the discrepancy between the calculated nodal thermal flux within an element and a continuous global thermal flux. This continuous thermal flux is calculated with the normal nodal averaging procedure.

The volume (result label VOLU) is used to calculate the energy error per element (result label SERR for the structural energy error and TERR for the thermal energy error). These energy errors, along with the appropriate energy, are then used to calculate the percent error in energy norm (SEPC for structural and TEPC for thermal). These percentages can be listed by the **PRERR (p. 1480)** command, retrieved by the ***GET (p. 797)** command (with labels SEPC and TEPC) for further calculations, and shown on the displacement display (**PLDISP (p. 1400)**), as applicable.

For structural analyses, the maximum absolute value of nodal stress variation of any stress component for any node of an element (result item SDSG) is also calculated. Similarly, for thermal gradient components, TDSG is calculated. Minimum and maximum result bounds considering the possible effect of discretization error will be shown on contour displays (**PLNSOL (p. 1425)**). For shell elements, the top surface location is used to produce a meaningful percentage value. SERR, TERR, SEPC, TEPC, SDSG, and TDSG will be updated whenever the nodal stresses or fluxes are recalculated.

If the energy error is a significant portion of the total energy, then the analysis should be repeated using a finer mesh to obtain a more accurate solution. The energy error is relative from problem to problem but will converge to a zero energy error as the mesh is refined.

The following element- and material-type limitations apply:

- Valid with most 2D solid, 3D solid, axisymmetric solid, or 3D shell elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The model should have only structural or thermal degrees of freedom.
- The analysis must be linear (for both material and geometry).
- Multi-material (for example, composite) elements are not valid.
- Transition regions from one material to another are not valid (that is, the entire model should consist of one material).
- Anisotropic materials (**TB (p. 1899)**,ANEL) are not considered.

Menu Paths

Main Menu>General Postproc>List Results>Percent Error

Utility Menu>List>Results>Percent Error

PRESOL, *Item*, *Comp*

Prints the solution results for elements.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label. For selected result output, specify SRES and see Table 220: PRESOL - Selected Result (SRES) Component Labels (p. 1489).

Comp

Component of the item (if required). Valid component labels are shown in the table below.

Notes

PRESOL (p. 1481) prints the solution results for the selected elements in the sorted sequence.

For example, **PRESOL (p. 1481)**,S prints the stress items SX, SY, SZ, SXY, SYZ, and SXZ for the node locations of the element. Various element results depend on the calculation method (**AVPRIN (p. 210)**).

Component results are in the global Cartesian coordinate directions unless transformed (**RSYS (p. 1639)**).

Shell elements print values at the top, then bottom of the element (or layer). If KEYOPT(8) = 2 (for SHELL181, SHELL208, SHELL209, SHELL281, or ELBOW290), the results are printed in the order TOP, BOT and then MID of each element, (or layer). The MID value is the actual value to the results file.

Items are listed as columns of a table versus element number. An exception occurs for item ELEM, which uses an element format (where all applicable line element results are listed per element) instead of a tabular format. An exception also occurs for the section results of beam- and pipe-based elements, which use a different fixed format and do not change with the **/FORMAT (p. 774)** specifications of the command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can issue **FORCE (p. 772)** to define which component of the nodal load is to be used (static, damping, inertia, or total).

To print items not available via **PRESOL (p. 1481)** (such as line element results), see **ETABLE (p. 687)** and **PRETAB (p. 1490)**.

For PowerGraphics (**/GRAPHICS (p. 866)**,POWER), results are listed only for the element surface. Items not supported by PowerGraphics are noted in Table 219: PRESOL - General Result Item and Component Labels (p. 1482).

The results printed by **PRESOL (p. 1481)** are unaffected by any requested nodal-averaged results (**OUTRES (p. 1336)**,NAR). For more information, see Nodal-Averaged Results in the *Element Reference*.

For *Item* = SRES, selected result components (**OSRESULT (p. 1327)**) are output. See Table 220: PRESOL - Selected Result (SRES) Component Labels (p. 1489).

Table 219: PRESOL - General Result Item and Component Labels

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
S	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) stresses.
	PRIN	Principal stresses (1, 2, 3), stress intensity (INT), and equivalent stress (EQV).
EPEL	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) elastic strains.
	PRIN	Principal elastic strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV).
EPTH	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) thermal strains.
	PRIN	Principal thermal strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV).
EPDI	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) diffusion strains.
	PRIN	Principal diffusion strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV).
EPPL	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) plastic strains.
	PRIN	Principal plastic strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV).
EPCR	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) creep strains.
	PRIN	Principal creep strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV).
EPSW		Swelling strain.
EPTO	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) total mechanical strains (EPEL + EPPL + EPCR).
	PRIN	Principal total mechanical strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV).
EPTT	COMP or blank	Component (X, Y, Z, XY, YZ, XZ) total mechanical, thermal, diffusion, and swelling strains (EPEL + EPPL + EPCR + EPTH + EPDI + EPSW).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
	PRIN	Principal total mechanical, diffusion, thermal, and swelling strains (1, 2, 3), strain intensity (INT), and equivalent strain (EQV).
NL		Nonlinear items (SEPL, SRAT, HPRES, EPEQ, CREQ, PSV, PLWK).
SEND	ELASTIC[a]	Elastic strain energy density. (For viscoelastic and sintering materials, the stored energy.)
	PLASTIC	Plastic strain energy density.
	CREEP	Creep strain energy density.
	DAMAGE	Damage strain energy density.
	VDAM[a]	Viscoelastic dissipation energy density.
	VREG	Visco-regularization strain energy density.
	DISS	Structural-thermal dissipation.
	ENTO	Total strain energy density (sum of ELASTIC, PLASTIC, and CREEP strain energy densities).
CDM	DMG	Damage variable.
	LM	Maximum previous strain energy for virgin material.
FAIL		Failure criteria for virgin material.[b][c] **Default components:** Maximum of all failure criteria defined at current location (MAX), maximum strain (EMAX), maximum stress (SMAX), Tsai-Wu Strength Index (TWSI), inverse of Tsai-Wu Strength Ratio Index (TWSR). **Other available components:** Hashin Fiber Failure (HFIB), Hashin Matrix Failure (HMAT), Puck Fiber Failure (PFIB), Puck Matrix Failure (PMAT), LaRc03 Fiber Failure (L3FB), LaRc03 Matrix Failure (L3MT), LaRc04 Fiber Failure (L4FB), LaRc04 Matrix Failure (L4MT), and any user-defined failure criteria (USR1 through USR9).[d] Issue **FCTYP (p. 740)** to activate or remove failure criteria.
PFC		Failure criteria based on the effective stresses in the damaged material. **Components:** Maximum of all failure criteria defined at current location (MAX), fiber tensile failure (FT), fiber compressive failure (FC), matrix tensile failure (MT), and matrix compressive (MC).
PDMG		Progressive damage parameters.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
		Components: Damage status (STAT, 0 = undamaged, 1 = damaged, 2 = complete damage), fiber tensile damage variable (FT), fiber compressive damage variable (FC), matrix tensile damage variable (MT), matrix compressive damage variable (MC), shear damage variable (S), energy dissipated per unit volume (SED), energy per unit volume due to viscous damping (SEDV).
FCMX		Maximum failure criterion over the entire element.[b][c] **Components:** Layer number where the maximum occurs (LAY), name of the maximum failure criterion (FC), and value of the maximum failure criterion (VAL).
SVAR	1,2,3, ... N	State variable.
GKS		Gasket component (X, XY, XZ) stress.
GKD		Gasket component (X, XY, XZ) total closure.
GKDI		Gasket component (X, XY, XZ) total inelastic closure.
GKTH		Gasket component (X, XY, XZ) thermal closure.
CONT		Contact items (STAT, PENE, PRES, SFRIC, STOT, SLIDE, GAP, FLUX, CNOS, FPRS). See component descriptions in **PLESOL (p. 1401)**.
TG		Component (X, Y, Z) thermal gradients and vector sum (SUM).[e]
TF		Component (X, Y, Z) thermal fluxes and vector sum (SUM).[e]
PG		Component (X, Y, Z) and vector sum (SUM) for velocity or energy density flux (room acoustics).
EF		Component (X, Y, Z) electric fields and vector sum (SUM).
D		Component (X, Y, Z) electric flux densities and vector sum (SUM).
H		Component (X, Y, Z) magnetic field intensities and vector sum (SUM).
B		Component (X, Y, Z) magnetic flux densities and vector sum (SUM).
CG		Component concentration gradient or vector sum.
DF		Component diffusion flux density or vector sum.
FMAG		Component (X, Y, Z) electromagnetic forces and vector sum (SUM).[b]
P		Poynting vector components (X, Y, Z) and sum (SUM).[b]
CG		Concentration gradient.
F		Component (X, Y, Z) structural forces.[b][f][g]
M		Component (X, Y, Z) structural moments.[f][b]

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
HEAT		Heat flow.[f][b]
FLOW		Fluid flow.[f]
AMPS		Current flow.[f][b]
CHRG		Charge.[f][b]
FLUX		Magnetic flux.[f][b]
CSG		Component (X, Y, Z) magnetic current segments.[f][b]
FORC		All available force items (F to CSG above). (10 maximum).[b]
RATE		Diffusion flow rate.
BFE	TEMP[h]	Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
ELEM		All applicable element results (available only for LINK180 and previous-generation structural line elements).[b]
SERR[i]		Structural error energy.[b]
SDSG[i]		Absolute value of the maximum variation of any nodal stress component.[b]
TERR[i]		Thermal error energy.[b]
TDSG[i]		Absolute value of the maximum variation of any nodal thermal gradient component.[b]
SENE		"Stiffness" energy or thermal heat dissipation. Same as TENE.[b]
STEN		Elemental energy dissipation due to stabilization.
TENE		Thermal heat dissipation or "stiffness" energy. Same as SENE.[b]
KENE		Kinetic energy.[b]
ASENE		Amplitude "stiffness" energy.[b]
PSENE		Peak "stiffness" energy.[b]
AKENE		Amplitude kinetic energy.[b]
PKENE		Peak kinetic energy.[b]
DENE		Damping energy.[b]
WEXT[j]		Work due to external load.[b]
AENE		Artificial energy due to hourglass control/drill stiffness or due to contact stabilization.[b]
JHEAT		Element Joule heat generation (coupled-field calculation).[b]
JS		Source current density for low-frequency magnetic analyses. Total current density (sum of conduction and displacement current densities) in low frequency electric analyses. Components (X, Y, Z) and vector sum (SUM).[b]
JT		Total measureable current density in low-frequency electromagnetic analyses. (Conduction current density in

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
		a low-frequency electric analysis.) Components (X, Y, Z) and vector sum (SUM).[b]
JC		Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).[b]
MRE		Magnetic Reynolds number.[b]
VOLU		Volume of volume element.[b]
CENT		Centroid X, Y, or Z location (based on shape function) in the active coordinate system.[b]
LOCI		Integration point location.
SMISC	*snum*	Element summable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description).[b]
NMISC	*snum*	Element nonsummable miscellaneous data value at sequence number *snum* (shown in the Output Data section of each element description).[b]
CAP		Material cap plasticity model only: Cohesion (C0); hydrostatic compaction yielding stress (X0); I1 at the transition point at which the shear and compaction envelopes intersect (K0); ZONE = 0: elastic state, ZONE = 1: compaction zone, ZONE = 2: shear zone, ZONE = 3: expansion zone; effective deviatoric plastic strain (DPLS); volume plastic strain (VPLS).
EDPC		Material EDP creep model only (not including the cap model): Equivalent creep stress (CSIG); equivalent creep strain (CSTR).
FICT	TEMP	Fictive temperature.
ESIG	COMP or blank	Components of Biot's effective stress.
	PRIN	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
	GPOR	Porosity due to void growth.
	NPOR	Porosity due to void nucleation.
FFLX	COMP	Fluid flow flux components in poromechanics.
FGRA	COMP	Fluid pore-pressure gradient components in poromechanics.
MENE		Acoustic potential energy.[b]
PMSV	COMP	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure CPT elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
FPIDX	TF01,SF01, TF02,SF02, TF03,SF03, TF04,SF04	Failure plane surface activity status for concrete and joint rock material models: 1 = yielded, 0 = not yielded. Tension and shear failure status are available for all four sets of failure planes.
YSIDX	TENS,SHEA	Yield surface activity status for Mohr-Coulomb, soil, concrete, and joint rock material models: 1 = yielded, 0 = not yielded.
NS	COMP	Nominal strain for hyperelastic material, reported in the current configuration (unaffected by **RSYS (p. 1639)**).
MPLA	DMAC,DMAX	Microplane damage, macroscopic and maximum values.
MPDP		Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).
DAMAGE		Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).
GDMG		Damage
IDIS		Structural-thermal dissipation rate
BKS	X, Y, Z, XY, YZ, XZ	Total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements.
BKS1, . . . ,BKS5	X, Y, Z, XY, YZ, XZ	Superimposed components of the total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements when more than one superimposed back-stress component is defined.
EPFR		Free strain in porous media
SNDI		Component (X, Y, Z) sound intensity and vector sum (SUM).[b]
FC1S	1,2,3,4,5,6	First set of six components of FCC crystal slip. Available for 3D elements only.
FC2S	1,2,3,4,5,6	Second set of six components of FCC crystal slip. Available for 3D elements only.
HC1S	1,2,3,4,5,6	Six components of HCP crystal slip on basal and prismatic systems. Available for 3D elements only.
HC2S	1,2,3,4,5,6	Six components of HCP crystal slip on pyramidal system. Available for 3D elements only.
HC3S	1,2,3,4,5,6	First set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC4S	1,2,3,4,5,6	Second set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC5S	1,2,3,4,5,6	Six components of HCP crystal slip on the second-order pyramidal system. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
BC1S	1,2,3,4,5,6	First set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC2S	1,2,3,4,5,6	Second set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC3S	1,2,3,4,5,6	First set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC4S	1,2,3,4,5,6	Second set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC5S	1,2,3,4,5,6	First set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC6S	1,2,3,4,5,6	Second set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC7S	1,2,3,4,5,6	Third set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC8S	1,2,3,4,5,6	Fourth set of six components of BCC slip on 123 plane. Available for 3D elements only.
FC1H	1,2,3,4,5,6	First set of six components of FCC crystal hardness. Available for 3D elements only.
FC2H	1,2,3,4,5,6	Second set of six components of FCC crystal hardness. Available for 3D elements only.
HC1H	1,2,3,4,5,6	Sixcomponents of HCP crystal hardness on basal and prismatic systems. Available for 3D elements.
HC2H	1,2,3,4,5,6	Six components of HCP crystal hardness on pyramidal system. Available for 3D elements only.
HC3H	1,2,3,4,5,6	First set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC4H	1,2,3,4,5,6	Second set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC5H	1,2,3,4,5,6	Six components of HCP crystal hardness on the second-order pyramidal system. Available for 3D elements only.
BC1H	1,2,3,4,5,6	First set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC2H	1,2,3,4,5,6	Second set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC3H	1,2,3,4,5,6	First set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC4H	1,2,3,4,5,6	Second set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC5H	1,2,3,4,5,6	First set of six components of BCC hardness on 123 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRESOL (p. 1481), *Item, Comp*		
Item	**Comp**	**Description**
BC6H	1,2,3,4,5,6	Second set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC7H	1,2,3,4,5,6	Third set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC8H	1,2,3,4,5,6	Fourth set of six components of BCC hardness on 123 plane. Available for 3D elements only.
XELG	1,2,3,45,6,EQV	Crystal Lagrangian strain in 11, 22, 33, 12, 23,13 directions and its equivalent. Available for 3D elements only.
SINT	RHO, ETA, SSTR, GRAIN	Sintering relative density, viscosity, sintering stress, and average grain size values.

[a] The results for this postprocessing SEND component are invalid for ELBOW290 if that element is used with viscoelastic or viscohyperelastic materials.

[b] Not supported by PowerGraphics.

[c] Works only if failure criteria are provided (**FC (p. 732)** and **TB (p. 1899)**).

[d] USR1 through USR9 require a failure-criteria routine.

[e] No vector sum is calculated for coupled pore-pressure-thermal (CPT*nnn*) elements.

[f] Use **FORCE (p. 772)** for type.

[g] Do not use **PRESOL (p. 1481)** to obtain contact forces for contact elements, as the force values reported may not be accurate for these elements. Use **ETABLE (p. 687)** instead.

[h] For reinforcing elements REINF264 and REINF265, issue **PRESOL (p. 1481)**,BFE,TEMP to print the intersection-point temperature of each member. You can also print intersection-point temperature gradients (**PRESOL (p. 1481)**,TG) and intersection-point heat flux (**PLESOL (p. 1401)**,TF). For higher-order reinforcing members (generated when using higher-order base elements), the midpoint values are not available for the reinforcing members.

[i] Some element- and material-type limitations apply. See **PRERR (p. 1480)**.

[j] WEXT is calculated for element-based loading only (and not for nodal-force loading). WEXT is stored on elements to which loading has been applied; if surface elements are added on top of other elements, for example, and pressure loading is applied to the surface elements, WEXT is available for the surface elements only.

Table 220: PRESOL - Selected Result (SRES) Component Labels

PRESOL (p. 1481),SRES,*Comp*	
Comp	**Description**
SVAR*n*	The *n*th state variable.
FLDUF0*n*	The *n*th user-defined field variable.
ItemComp (concatenated *Item* + *Comp* label) from Table 208: OSRESULT - Item and Component Labels (p. 1328)[a]	See the table (p. 1328) for the combined item and component description.

[a] See Example 10: Specifying and Retrieving Selected Result Output (p. 1329).

Menu Paths

Main Menu>General Postproc>List Results>Element Solution

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Utility Menu>List>Results>Element Solution

PRETAB, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*, *Lab7*, *Lab8*, *Lab9*

Prints the element table items.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab1*, *Lab2*, *Lab3*, . . . , *Lab9

Print selected items. Valid labels are (blank) or any label as specified with the **ETABLE (p. 687)** command. Convenience labels may be used for *Lab1* to select groups of labels (10 labels maximum): GRP1 for first 10 stored items; GRP2 for items 11 to 20; GRP3 for items 21 to 30; GRP4 for items 31 to 40; GRP5 for items 41 to 50. Enter **ETABLE (p. 687)**,STAT command to list stored item order. If all labels are blank, print first 10 stored items (GRP1).

Notes

Prints the items stored in the table defined with the **ETABLE (p. 687)** command. Item values will be listed for the selected elements in the sorted sequence (**ESORT (p. 680)**). The **FORCE (p. 772)** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total).

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

Menu Paths

Main Menu>General Postproc>Element Table>List Elem Table

Main Menu>General Postproc>List Results>Elem Table Data

Utility Menu>List>Results>Element Table Data

PRFAR, *Lab*, *Option*, *VAR1B*, *VAR1E*, *NVAR1*, *VAR2B*, *VAR2E*, *NVAR2*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *LDSTEP*, *SUBSTEP*, *FREQB*, *FREQE*, *PrintType*, *LogOpt*

Prints acoustic far field parameters.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Parameters to print:

PRES

Acoustic parameters

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PROT

Acoustic parameters with the y-axis rotated extrusion (not valid for 2D elements)

PLAT

Acoustic parameters radiated by a vibrating structural panel (not valid for 2D elements)

Option

Print option, based on the specified print parameter type:

Lab	*Option*
PRES, PROT, *or* PLAT	SUMC -- Maximum pressure (default) PHSC -- Pressure phase angle SPLC -- Sound pressure level SPAC – A-weighted sound pressure level (dBA) DGCT -- Acoustic directivity PSCT -- Maximum scattered pressure TSCT -- Target strength PWL -- Sound power level

The arguments *VAR1B*, *VAR1E*, *NVAR1*, *VAR2B*, *VAR2E*, *NVAR2*, and *VAL1* are used when *Option* = SUMC, PHSC, SPLC, SPAC, DGCT, PSCT, or TSCT:

VAR1B, VAR1E

Starting and ending values for the first variable associated with *PrintType* as described below.

When *PrintType* = blank (default) or SPHR: Starting and ending phi (ϕ) angles (in degrees) in the spherical coordinate system. Defaults to 0.

When *PrintType* = PLXY: Starting and ending x value in the Cartesian coordinate system. Defaults to 0.

When *PrintType* = PLYZ: Starting and ending y value in the Cartesian coordinate system. Defaults to 0.

When *PrintType* = PLXZ: Starting and ending x value in the Cartesian coordinate system. Defaults to 0.

NVAR1

Number of divisions between the starting and ending *VAR1* values for data computations. Defaults to 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAR2B, VAR2E

Starting and ending values for the second variable associated with *PrintType* as described below.

When *PrintType* = blank (default) or SPHR: Starting and ending theta (θ) angles (in degrees) in the spherical coordinate system. Defaults to 0 for a 3D model and 90 for a 2D extrusion model.

When *PrintType* = PLXY: Starting and ending y value in the Cartesian coordinate system. Defaults to 0.

When *PrintType* = PLYZ: Starting and ending z value in the Cartesian coordinate system. Defaults to 0.

When *PrintType* = PLXZ: Starting and ending z value in the Cartesian coordinate system. Defaults to 0.

NVAR2

Number of divisions between the starting and ending *VAR2* values for data computations. Defaults to 0.

VAL1

VAL1 is additional input. Its meaning depends on the *PrintType* argument as described below.

When *PrintType* = blank (default) or SPHR: Radius of the sphere surface.

When *PrintType* = PLXY: Fixed z value for an X-Y plane in the Cartesian coordinate system. Defaults to 0.

When *PrintType* = PLYZ: Fixed x value for a Y-Z plane in the Cartesian coordinate system. Defaults to 0.

When *PrintType* = PLXZ: Fixed y value for an X-Z plane in the Cartesian coordinate system, Defaults to 0.

The arguments *VAL2* through *VAL5* are additional input used with certain *Lab* and *Option* settings as described below.

VAL2

When *Option* = SPLC or SPAC: Reference rms sound pressure. Defaults to $2x10^{-5}$ Pa.

When *Option* = PWL: Reference sound power. Defaults to $1x10^{-12}$ watts.

VAL3

When *Lab* = PRES: Thickness of 2D model extrusion in the z direction (no default).

When *Lab* = PROT: Angle of the y-axis rotated extrusion model (no default)

VAL4

When *Lab* = PLAT: Mass density of acoustic fluid.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL5

When *Lab* = PLAT: Sound speed in acoustic fluid.

LDSTEP

Specified load step. Defaults to the load step number specified on the **SET (p. 1724)** command, or defaults to 1 if **SET (p. 1724)** has not been issued.

n

Load step number.

ALL

All load steps.

SUBSTEP

Specified substep. Defaults to the substep number specified on the **SET (p. 1724)** command, or defaults to ALL (all substeps at the specified load step) if **SET (p. 1724)** has not been issued.

n

Substep number.

ALL

All substeps.

FREQB

Frequency value representing one of the following:

- Beginning frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If a *SUBSTEP* value is specified, *FREQB* is invalid.
- Central frequency of octave bands, used when *LogOpt* = OB1, OB2, OB3, OB6, OB12, or OB24 and *FREQE* is blank.

FREQE

Ending frequency of the frequency range (*FREQB* to *FREQE*) for the defined load step(s) and substeps (*SUBSTEP* = ALL). If blank, *FREQE* is set to *FREQB*. If a *SUBSTEP* value is specified, *FREQE* is invalid.

PrintType

Print out far-field parameters on a plane or sphere (used when *Option* = SUMC, PHSC, SPLC, SPAC, DGCT, PSCT, or TSCT). No default.

PLXY

On an X-Y plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PLYZ

On a Y-Z plane.

PLXZ

On an X-Z plane.

SPHR

On a sphere surface.

LogOpt

Octave bands (used only when Option = SPLC, SPAC, or PWL):

OB0

Narrow bands (default).

OB1

Octave bands.

OB2

1/2 octave bands.

OB3

1/3 octave bands.

OB6

1/6 octave bands.

OB12

1/12 octave bands.

OB24

1/24 octave bands.

Notes

The **PRFAR (p. 1490)** command prints pressure far fields and far field parameters as determined by the equivalent source principle. Use this command to print pressure and acoustic parameters. See the **HFSYM (p. 905)** command for the model symmetry and the **HFANG (p. 904)** command for spatial radiation angles.

Printing acoustic parameters radiated by a vibrating structural panel (*Lab* = PLAT) is supported by elements SOLID185, SOLID186, SOLID187, SHELL181, SHELL281, and SOLSH190. The vibration surface of the panel must be flagged by the **SF (p. 1733)**,MXWF command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

PRI2, *P51X*, *Z1*, *Z2*

Creates a polygonal area or a prism volume by vertices (GUI).

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Creates a polygonal area or a prism volume using the vertices as input. This is a command generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if graphical picking is used.

This command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

For polygons, **PRI2 (p. 1495)** appears in the log file as **PRI2 (p. 1495)**,*P51X*,0.0,0.0, preceded by **FITEM (p. 758)** commands defining the vertices (in global Cartesian coordinates).

For prisms, **PRI2 (p. 1495)** appears in the log file as **PRI2 (p. 1495)**,*P51X*, preceded by **FITEM (p. 758)** commands defining the vertices and the Z-end of the prism.

See **RPOLY (p. 1616)**, **POLY (p. 1460)**, **RPRISM (p. 1618)**, **PRISM (p. 1497)**, and **RPR4 (p. 1617)** for other ways to create polygons and prisms.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Vertices

Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Vertices

---

PRIM

Specifies "Solid model primitives" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.LOG`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Solid Model

PRINT

Specifies "Print settings" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)
POST26 (p. 58): Status (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>List Results

Utility Menu>List>Status>TimeHist Postproc>List

---

*PRINT,*Matrix*,*Fname*

Prints the matrix values to a file.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Matrix

Name of matrix or vector to print. Must be specified.

Fname

File name (case-sensitive, 32-character maximum). If blank, matrix is written to the output file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The matrix may be a dense matrix (***DMAT (p. 551)**), a sparse matrix (***SMAT (p. 1801)**), or a vector (***VEC (p. 2085)**). Only the non-zero entries of the matrix are printed.

Menu Paths

This command cannot be accessed from a menu.

PRISM, *Z1*, *Z2*

Creates a prism volume based on working plane coordinate pairs.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Z1, Z2

Working plane Z coordinates of the top and bottom of the prism.

Notes

Defines a prism volume based on the working plane. The top and bottom areas will each be defined with NPT keypoints and NPT lines, where NPT (must be at least 3) is the number of coordinate pairs defined with **PTXY (p. 1556)** command. Also, a line will be defined between each point pair on the top and bottom face. See the **RPRISM (p. 1618)** and **RPR4 (p. 1617)** commands for other ways to create prisms.

Menu Paths

This command cannot be accessed from a menu.

PRITER

Prints solution summary data.

POST1 (p. 51): Listing (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Prints solution summary data (such as time step size, number of equilibrium iterations, convergence values, etc.) from a static or full transient analysis. All other analyses print zeros for the data.

Menu Paths

Main Menu>General Postproc>List Results>Iteration Summry

Utility Menu>List>Results>Iteration Summry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRJSOL, *Item*, *Comp*

Prints joint element output.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item. Some items also require a component label.

DISP

Relative displacements.

ROT

Relative rotations.

VEL

Relative linear velocities.

OMG

Relative angular velocities.

ACC

Relative linear accelerations.

DMG

Relative angular accelerations.

SMISC

Summable miscellaneous quantities.

Comp

Component of the item (if required). For *Item* = DISP, ROT, VEL, OMG, ACC, and DMG, enter the direction label, X, Y, or Z. For *Item* = SMISC, enter a valid number.

Notes

Prints element output for the MPC184 joint element. The joint element quantities printed are the values for the free or unconstrained relative degrees of freedom.

Only **PRJSOL (p. 1498)**,SMISC is available in linear, modal, and linear perturbation analyses.

Only **PRJSOL (p. 1498)**,SMISC is available for penalty-based joint elements.

This command is valid in POST1 only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

PRMC, *LSTEP*, *SBSTEP*, *TIMFRQ*, *KIMG*, *HIbeg*, *HIend*, *Matrix*

Prints the modal coordinates from a mode-superposition solution.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSTEP, SBSTEP

Print the solution identified as load step *LSTEP* and substep *SBSTEP*.

TIMFRQ

As an alternative to *LSTEP* and *SBSTEP*, print the solution at the time value *TIMFRQ* (for **ANTYPE (p. 162)**,TRANS) or frequency value *TIMFRQ* (for **ANTYPE (p. 162)**,HARMIC). *LSTEP* and *SBSTEP* should be left blank.

KIMG

Key for printing real or imaginary solution. Valid only for **ANTYPE (p. 162)**,HARMIC.

0 (or blank)

Print the real solution (default).

1

Print the imaginary solution.

2

Print the amplitude.

HIbeg, HIend

For cyclic symmetry solutions, print the solutions in the harmonic index solution range *HIbeg* to *HIend*. Defaults to all harmonic indices (all modes).

Matrix

Create an APDL Math dense matrix with the name entered on this field (up to 32 characters; for nomenclature guidelines see Guidelines for Parameter Names in the *Ansys Parametric Design Language Guide*). The matrix contains the modal coordinates. By default (*Matrix* = blank), no APDL Math matrix is created.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

PRMC (p. 1499) prints the modal coordinates (the factors which modes may be multiplied by to obtain their contribution to the response) at a certain time point (transient analyses) or frequency point (harmonic analyses).

The printout contains four columns: the mode number (labelled MODE), the modal frequency (labelled FREQ), the modal coordinate or mode multiplier (labelled MULT), and the normalized modal coordinate (labelled NORM). The normalized modal coordinate is the ratio of absolute value of the mode multiplier divided by the sum of the absolute values of all multipliers listed (at a solution time/frequency and harmonic index). It may be useful for identifying the dominant modes. Maximum values of each column are also listed at the end of each report.

By default, the real part of the modal coordinate values are printed even if the modal coordinates are complex.

When *Matrix* is specified, an APDL Math dense matrix similar to the one created with the ***DMAT (p. 551)** command is created. If **PRMC (p. 1499)** is issued multiple times with the same name entered on *Matrix* or if a matrix with the specified name already exists, the matrix is overwritten. This matrix contains four to five columns depending on the analysis. The first four columns are the ones printed by **PRMC (p. 1499)**. The fifth column contains the harmonic index for cyclic analysis only. This matrix can then be used in APDL Math data processing and file handling (See APDL Math in the *Ansys Parametric Design Language Guide*). For instance, ***EXPORT (p. 716)** can be issued to export the **PRMC (p. 1499)** data to a `.csv` file.

For transient analyses, a `.rdsp` file must be available. For harmonic analyses, a `.rfrq` file must be available. The content of these files depends on the **OUTRES (p. 1336)** command settings. Note that the default for mode-superposition transient analysis is to write the reduced displacement file every 4th substep. For more information, see Command Default (p. 1341) in the **OUTRES (p. 1336)** command description.

For a cyclic harmonic mode-superposition analysis, use the **CYCFILES (p. 457)** command to identify the `.rfrq` and modal `.rst` files. For other analyses, use the **FILE (p. 752)** command to specify the `.rdsp` or `.rfrq` file.

This information can also be obtained from the optional `Jobname.mcf` text file (see the **TRNOPT (p. 2017)** and **HROPT (p. 912)** commands), and it can be plotted using the **PLMC (p. 1421)** command. For more information on modal coordinates, see Mode-Superposition Method in the *Mechanical APDL Theory Reference*

Example Usage

```
/POST1
FILE,,rdsp                    ! Specify Jobname.rdsp file from a previous MSUP transient analysis
! Print modal coordinates from the second loadstep and fourth substep
PRMC,2,4,,,,,MAT              ! also create an APDL Math matrix called MAT
*EXPORT,MAT,CSV,PRMCFILE.CSV  ! Export MAT to a .csv file
```

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRNEAR, *Lab*, *Opt*, *KCN*, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*

Prints the pressure in the near zone exterior to the equivalent source surface.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Print the maximum pressure or sound pressure level:

POINT

at the point (x,y,z)

SPHERE

on the spherical structure

PATH

along the path

Opt

PSUM

Maximum complex pressure for acoustics.

PHAS

Phase angle of complex pressure for acoustics.

SPL

Sound pressure level for acoustics.

SPLA

A-weighted sound pressure level for acoustics (dBA).

KCN

KCN is the coordinate system reference number. It may be 0 (Cartesian) or any previously defined local coordinate system number (>10). Defaults to 0.

VAL1, VAL2, VAL3, . . . , VAL9

For *Lab* = POINT:

VAL1

x coordinate value

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL2

y coordinate value

VAL3

z coordinate value

VAL4 - VAL8

not used

VAL9

Thickness of model in z direction (defaults to 0).

For *LAB* = SPHERE:

VAL1

Radius of spherical surface in spherical coordinate system.

VAL2

Starting ϕ angle (degree) in the spherical coordinate system. Defaults to 0.

VAL3

Ending ϕ angle (degree) in the spherical coordinate system. Defaults to 0.

VAL4

Number of divisions between the starting and ending ϕ angles for data computations. Defaults to 0.

VAL5

Starting θ angle (degrees) in the spherical coordinate system. Defaults to 0 in 3D and 90 in 2D extension.

VAL6

Ending θ angle (degrees) in the spherical coordinate system. Defaults to 0 in 3D and 90 in 2D extension.

VAL7

Number of divisions between the starting and ending θ angles for data computations. Defaults to 0.

VAL8

Reference rms sound pressure. Defaults to $2x10^{-5}$ Pa.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL9

Thickness of 2D model extension in z direction (defaults to 0).

For *Lab* = **PATH (p. 1354)**, **PRNEAR (p. 1501)** computes the pressure for the path data points for the path currently defined by the **PATH (p. 1354)** and **PPATH (p. 1463)** commands.

Notes

The command uses the equivalent source principle to calculate the pressure in the near zone exterior to the equivalent source surface (flagged with the Maxwell surface flag in the preprocessor) for one of the following locations:

- A point X, Y, Z in the KCN coordinate system
- A spherical surface in the KCN coordinate system
- A path defined by the **PATH (p. 1354)** and **PPATH (p. 1463)** commands

To list the pressure results for a path, use the **PRPATH (p. 1517)** command. See **HFSYM (p. 905)** command for the model symmetry.

To retrieve saved equivalent source data, issue the **SET (p. 1724)**,*Lstep*,*Sbstep*,,REAL command.

Menu Paths

This command cannot be accessed from a menu.

PRNLD,*Lab*,*TOL*,*Item*

Prints the summed element nodal loads.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are:

Structural force labels: FX, FY or FZ (forces); F (FX, FY and FZ); MX, MY or MZ (moments); M (MX, MY and MZ).

Thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow).

Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid "forces"); VF (VFX, VFY and VFZ).

Electric force labels: AMPS (current flow); CHRG (charge); CURT (current); VLTG (voltage drop).

Magnetic force labels: FLUX (magnetic flux); CSGZ (magnetic current segment).

Diffusion label: RATE (diffusion flow rate).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TOL

Tolerance value about zero within which loads are not printed, as follows:

> 0

Relative tolerance about zero within which loads are not printed. In this case, the tolerance is *TOL* * *Load*, where *Load* is the absolute value of the maximum load on the selected nodes.

0

Print *all* nodal loads.

> 0

Absolute tolerance about zero within which loads are not printed.
Defaults to 1.0E-9 times the absolute value of the maximum load on the selected nodes.

ITEM

Selected set of nodes.

(blank)

Prints the summed element nodal loads for all selected nodes (default), excluding contact elements.

CONT

Prints the summed element nodal loads for contact nodes only.

BOTH

Prints the summed element nodal loads for all selected nodes, including contact nodes.

Notes

Prints the summed element nodal loads (forces, moments, heat flows, flux, etc.) for the selected nodes in the sorted sequence. Results are in the global Cartesian coordinate directions unless transformed (**RSYS (p. 1639)**). Zero values (within a tolerance range) are not printed. Loads applied to a constrained degree of freedom are not included. The **FORCE (p. 772)** command can be used to define which component of the nodal load is to be used (static, damping, inertia, or total).

By default, **PRNLD (p. 1503)** excludes elements TARGE169 - CONTA177. Setting *ITEM* = CONT will only account for nodal forces on selected contact elements (CONTA172, CONTA174, CONTA175, and CONTA177). Setting *ITEM* = BOTH will account for nodal forces on all selected nodes, including contact nodes.

Using PRNLD (p. 1503) in a Spectrum or PSD Analysis (ANTYPE (p. 162), SPECTR)

When using **PRNLD (p. 1503)** in a spectrum analysis (after the combination file has been input through **/INPUT (p. 948)**,,MCOM and when **SPOPT (p. 1837)** has not been issued with *Elcalc* = YES during the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

spectrum analysis), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS (p. 1639)** does not apply. The forces are in the nodal coordinate systems, and the vector sum is always printed in the global coordinate system.

The spectrum analysis summation is available when the element results are written to the mode file, `Jobname.mode` (*MSUPkey* = Yes on the **MXPAND (p. 1203)** command).

Because modal displacements cannot be used to calculate contact element nodal forces, *ITEM* does not apply to spectrum and PSD analyses.

Menu Paths

Main Menu>General Postproc>List Results>Nodal Loads

Utility Menu>List>Results>Nodal Loads

PRNSOL, *Item*, *Comp*, --, --, --, *Avg*, *DataKey*

Prints nodal solution results.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label. For selected result output, specify SRES and see Table 222: PRNSOL - Selected Result (SRES) Component Labels (p. 1514).

Comp

Component of the item (if required). Valid component labels are shown in the table below. Default = COMP.

--, --, --

Unused fields.

Avg

Specifies whether random acoustic results are averaged. Valid only for *Item* = U and PRES.

(blank)

No averaging (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AVG

Print averaged results for random acoustics.

DataKey

Key to specify which data is printed:

AUTO

Nodal-averaged results are used if available; otherwise, the element-based data is used, if available. (Default.)

ESOL

Only element-based results are used. If they are not available, the command is ignored.

NAR

Only nodal-averaged results are used. If they are not available, the command is ignored.

Notes

Prints the nodal solution results for the selected nodes in the sorted sequence. For reinforcing elements (REINF*nnn*), results are printed at intersection points of reinforcing elements and base elements.

For example, **PRNSOL (p. 1505)**,U,X prints the X component of displacement vector U (that is, the UX degree of freedom).

Component results are in the global Cartesian coordinate directions unless transformed (**RSYS (p. 1639)**).

Various element results also depend upon the recalculation method and the selected results location (**AVPRIN (p. 210)**, **RSYS (p. 1639)**, **LAYER (p. 1015)**, **SHELL (p. 1784)**, and **NSEL (p. 1266)**).

If **LAYER (p. 1015)** is issued, the resulting output is listed in full graphics mode (**/GRAPHICS (p. 866)**,FULL).

You can define which component of the nodal load (static, damping, inertia, or total) should be used (**FORCE (p. 772)**).

PowerGraphics can affect your nodal solution listings. For PowerGraphics (**/GRAPHICS (p. 866)**,POWER), results are listed for the model exterior surfaces only.

When shell element types are present, results are output on a surface-by-surface basis. For shell elements (such as SHELL181 or SHELL281), and for ELBOW290, printed output is for both the top and bottom surfaces. For solid elements such as SOLID185, the output is averaged for each surface and printed as follows:

- **Node at a vertex:** Three lines are output (one printed line for each surface).
- **Node on an edge:** Two lines are output (one printed line for each surface).
- **Nodes on a face:** One value is output.
- **Nodes interior to the volume:** No printed values are output.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a node is common to more than one element, or if a geometric discontinuity exists, several conflicting listings may result. For example, a corner node incorporating results from solid elements and shell elements could yield as many as nine different results; the printed output would be averages at the top and bottom for the three shell surfaces plus averages at the three surfaces for the solid, for a total of nine lines of output. The program does not average result listings across geometric discontinuities when shell element types are present. It is important to analyze the listings at discontinuities to ascertain the significance of each set of data.

When only reinforcing elements (REINF*nnn*) are selected, results are listed for intersection points of reinforcing elements and base elements. Prints include coordinates of intersection points in global Cartesian coordinate system and results. Results are interpolated from the results of base elements. If a point is common to more than one reinforcing element, or reinforcing member within one reinforcing element, averaged results are printed. Prints also include minimum and maximum values.

The printed output for full graphics (**/GRAPHICS (p. 866)**,FULL) averages results at the node. For shell elements, the default for display is TOP so that the results for the top of the shell are averaged with the other elements attached to that node.

If **NSORT (p. 1283)**, **ESORT (p. 680)** or **/ESHAPE (p. 665)** is issued with PowerGraphics enabled (**/GRAPHICS (p. 866)**,POWER), **PRNSOL (p. 1505)** behaves as though full graphics mode is enabled (**/GRAPHICS (p. 866)**,FULL).

Items not supported by PowerGraphics are noted in Table 221: PRNSOL - General Result Item and Component Labels (p. 1507).

For *Item* = SRES, selected result component (**OSRESULT (p. 1327)**) values are output. See Table 222: PRNSOL - Selected Result (SRES) Component Labels (p. 1514).

To print midside nodes, first issue **/EFACET (p. 602)**,2.

To learn more about the specific behaviors of **PRNSOL (p. 1505)** in a cyclic symmetry analysis and printing results for nodes at cyclic edges, see Using the /CYCEXPAND Command in the *Cyclic Symmetry Analysis Guide*.

If nodal-averaged results (**OUTRES (p. 1336)**,NAR or another nodal-averaged label) are in the database, then **PRNSOL (p. 1505)** uses the nodal-averaged data for the applicable items (S, EPEL, EPPL, EPCR, EPTH, EPSW) by default. You can change this behavior via the *DataKey* argument. Keep these points in mind when using nodal-averaged results:

- The **LAYER (p. 1015)** and **RSYS (p. 1639)**,SOLU commands are not valid with nodal-averaged results. If these commands are used, the element solution is printed instead if applicable.
- Issuing **ESEL (p. 661)** before printing nodal-averaged results has no effect on the output.
- PowerGraphics is supported. The output is equivalent to the full model graphics output, but only the appropriate surface nodes are printed. See Postprocessing Nodal-Averaged Results in the *Element Reference* for details.

Table 221: PRNSOL - General Result Item and Component Labels

General Item and Component Labels **PRNSOL (p. 1505)**, *Item*, *Comp*		
Item	**Comp**	**Description**
Valid item and component labels for nodal degree of freedom results are:		

General Item and Component Labels PRNSOL (p. 1505), *Item, Comp*		
Item	**Comp**	**Description**
U	X, Y, Z	X, Y, or Z structural displacement.
	COMP	X, Y, and Z structural displacements and vector sum.
ROT	X, Y, Z	X, Y, or Z structural rotation.
	COMP	X, Y, and Z structural rotations and vector sum.
TEMP[a]		Temperature.
PRES		Pressure.
VOLT		Electric potential.
GFV1, GFV2, GFV3		Nonlocal field values 1, 2, and 3.
MAG		Magnetic scalar potential.
CONC		Concentration.[b]
V	X, Y, Z	X, Y, or Z fluid velocity in a fluid analysis.
	COMP	X, Y, and Z fluid velocity and vector sum in a fluid analysis.
A	X, Y, Z	X, Y, or Z magnetic vector potential in an electromagnetic analysis.
	COMP	X, Y, and Z magnetic vector potential and vector sum in an electromagnetic analysis.
VEL	X, Y, Z	X, Y, or Z velocity in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
	COMP	X, Y, and Z velocity and vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
ACC	X, Y, Z	X, Y, or Z acceleration in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
	COMP	X, Y, and Z acceleration and vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
OMG	X, Y, Z	X, Y, or Z rotational velocity in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
	COMP	X, Y, and Z rotational velocity and vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
DMG	X, Y, Z	X, Y, or Z rotational acceleration in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
	COMP	X, Y, and Z rotational acceleration and vector sum in a structural transient dynamic analysis (**ANTYPE (p. 162)**,TRANS).
CURR		Current.
EMF		Electromotive force drop.
DOF		All available degree of freedom labels (10 maximum).
FICT	TEMP	Fictive temperature.
SPL		Sound pressure level.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRNSOL (p. 1505), *Item, Comp*		
Item	**Comp**	**Description**
SPLA		A-weighted sound pressure level (dBA).
VNS		Normal velocity on the structural surface.[c]
ENKE		Acoustic diffusion energy density
Valid item and component labels for element results are:		
S	COMP	X, Y, Z, XY, YZ, and XZ component stresses.[d]
	PRIN	S1, S2, S3 principal stresses, SINT stress intensity, and SEQV equivalent stress.[d]
EPEL	COMP	Component elastic strains.[d]
	PRIN	Principal elastic strains, strain intensity, and equivalent strain.[d]
	FAIL	Maximum Strain Failure Criteria.[b][e]
EPTH	COMP	Component thermal strains.[d]
	PRIN	Principal thermal strains, strain intensity, and equivalent strain.[d]
EPDI	COMP	Component diffusion strains.
	PRIN	Principal diffusion strains, strain intensity, and equivalent strain.
EPPL	COMP	Component plastic strains.[d]
	PRIN	Principal plastic strains, strain intensity, and equivalent strain.[d]
EPCR	COMP	Component creep strains.[d]
	PRIN	Principal creep strains, strain intensity, and equivalent strain.[d]
EPSW		Swelling strain.[d]
EPTO	COMP	Component total mechanical strains (EPEL + EPPL + EPCR).
	PRIN	Principal total mechanical strains, strain intensity, and equivalent strain.
EPTT	COMP	Component total mechanical, thermal, diffusion, and swelling strains (EPEL + EPPL + EPCR + EPTH + EPDI + EPSW).
	PRIN	Principal total mechanical, thermal, diffusion, and swelling strains, strain intensity, and equivalent strain.
NL		Nonlinear items (SEPL, SRAT, HPRES, EPEQ, CREQ, PSV, PLWK).
SEND	ELASTIC[f]	Elastic strain energy density. (For viscoelastic and sintering materials, the stored energy.)
	PLASTIC	Plastic strain energy density.
	CREEP	Creep strain energy density.
	DAMAGE	Damage strain energy density.
	VDAM[f]	Viscoelastic dissipation energy density.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRNSOL (p. 1505), *Item, Comp*		
Item	**Comp**	**Description**
	VREG	Visco-regularization strain energy density.
	DISS	Structural-thermal dissipation.
	ENTO	Total strain energy density (sum of ELASTIC, PLASTIC, and CREEP strain energy densities).
CDM	DMG	Damage variable.
	LM	Maximum previous strain energy for virgin material.
FAIL		Failure criteria.[b][e] **Default components:** Maximum of all failure criteria defined at current location (MAX), maximum strain (EMAX), maximum stress (SMAX), Tsai-Wu Strength Index (TWSI), inverse of Tsai-Wu Strength Ratio Index (TWSR). **Other available components:** Other available components: Hashin Fiber Failure (HFIB), Hashin Matrix Failure (HMAT), Puck Fiber Failure (PFIB), Puck Matrix Failure (PMAT), LaRc03 Fiber Failure (L3FB), LaRc03 Matrix Failure (L3MT), LaRc04 Fiber Failure (L4FB), LaRc04 Matrix Failure (L4MT), and any user-defined failure criteria (USR1 through USR9).[g] Issue **FCTYP (p. 740)** to activate or remove failure criteria.
PFC		Failure criteria based on the effective stresses in the damaged material. **Components:** Maximum of all failure criteria defined at current location (MAX), fiber tensile failure (FT), fiber compressive failure (FC), matrix tensile failure (MT), and matrix compressive (MC).
PDMG		Progressive damage parameters. **Components:** Damage status (STAT, 0 = undamaged, 1 = damaged, 2 = complete damage), fiber tensile damage variable (FT), fiber compressive damage variable (FC), matrix tensile damage variable (MT), matrix compressive damage variable (MC), shear damage variable (S), energy dissipated per unit volume (SED), energy per unit volume due to viscous damping (SEDV).
SVAR[h]	1, 2, 3, ... N	State variable.
GKS	COMP	X, XY, XZ component gasket stress.
GKD	COMP	X, XY, XZ component gasket total closure.
GKDI	COMP	X, XY, XZ component gasket total inelastic closure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRNSOL (p. 1505), *Item, Comp*		
Item	**Comp**	**Description**
GKTH	COMP	X, XY, XZ component thermal closure.
SS	X, XY, XZ	Interface traction (stress).
SD	X, XY, XZ	Interface separation.
CONT		Contact items (STAT[i], PENE, PRES, SFRIC, STOT, SLIDE, GAP, FLUX, CNOS, FPRS). See component descriptions in PLNSOL (p. 1425).
TG	COMP	Component thermal gradients and vector sum.[j]
TF	COMP	Component thermal fluxes and vector sum.[j]
PG	COMP	Components and vector sum for velocity or energy density flux (room acoustics).
EF	COMP	Component electric fields and vector sum.
D	COMP	Component electric flux densities and vector sum.
H	COMP	Component magnetic field intensities and vector sum.
B	COMP	Component magnetic flux densities and vector sum.
CG	COMP	Component concentration gradient or vector sum.
DF	COMP	Component diffusion flux density or vector sum.
FMAG	COMP	Component electromagnetic forces and vector sum.[b]
JC	COMP	Conduction current density for elements that support conduction current calculation. Components (X, Y, Z) and vector sum (SUM).[b]
BFE		Body temperatures (calculated from applied temperatures) as used in solution (area and volume elements only).
CAP		Material cap plasticity model only: Cohesion (C0); hydrostatic compaction yielding stress (X0); I1 at the transition point at which the shear and compaction envelopes intersect (K0); ZONE = 0: elastic state, ZONE = 1: compaction zone, ZONE = 2: shear zone, ZONE = 3: expansion zone; effective deviatoric plastic strain (DPLS); volume plastic strain (VPLS).
EDPC		Material EDP creep model only (not including the cap model): Equivalent creep stress (CSIG); equivalent creep strain (CSTR).
ESIG	COMP or blank	Components of Biot's effective stress.
	PRIN	Principal stresses of Biot's effective stress.
	INT	Stress intensity of Biot's effective stress.
	EQV	Equivalent stress of Biot's effective stress.
DPAR	TPOR	Total porosity (Gurson material model).
	GPOR	Porosity due to void growth.
	NPOR	Porosity due to void nucleation.
FFLX	COMP	Fluid flow flux in poromechanics.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRNSOL (p. 1505), *Item, Comp*		
Item	**Comp**	**Description**
FGRA	COMP	Fluid pore pressure gradient components in poromechanics.
PMSV	COMP	Void volume ratio, pore pressure, degree of saturation, and relative permeability for coupled pore-pressure CPT elements.
NS	COMP	Nominal strain for hyperelastic material, reported in the current configuration (unaffected by **RSYS (p. 1639)**).
MPLA	DMAC, DMAX	Microplane damage, macroscopic and maximum values.
MPDP		Microplane homogenized total, tension, and compression damages (TOTA, TENS, COMP), and split weight factor (RW).
DAMAGE		Damage in directions 1, 2, 3 (1, 2, 3) and the maximum damage (MAX).
GDMG		Damage
IDIS		Structural-thermal dissipation rate
BKS	X, Y, Z, XY, YZ, XZ	Total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements.
BKS1, . . . ,BKS5	X, Y, Z, XY, YZ, XZ	Superimposed components of the total nonlinear kinematic backstress reported in the current configuration (unaffected by **RSYS (p. 1639)**). Available for 3D, plane strain, and axisymmetric elements when more than one superimposed back-stress component is defined.
EPFR		Free strain in porous media
SNDI	COMP	Component sound intensity and vector sum.[b]
FC1S	1,2,3,4,5,6	First set of six components of FCC crystal slip. Available for 3D elements only.
FC2S	1,2,3,4,5,6	Second set of six components of FCC crystal slip. Available for 3D elements only.
HC1S	1,2,3,4,5,6	Six components of HCP crystal slip on basal and prismatic systems. Available for 3D elements only.
HC2S	1,2,3,4,5,6	Six components of HCP crystal slip on pyramidal system. Available for 3D elements only.
HC3S	1,2,3,4,5,6	First set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC4S	1,2,3,4,5,6	Second set of six components of HCP crystal slip on the first-order pyramidal system. Available for 3D elements only.
HC5S	1,2,3,4,5,6	Six components of HCP crystal slip on the second-order pyramidal system. Available for 3D elements only.
BC1S	1,2,3,4,5,6	First set of six components of BCC slip on 111 plane. Available for 3D elements only.
BC2S	1,2,3,4,5,6	Second set of six components of BCC slip on 111 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRNSOL (p. 1505), *Item*, *Comp*		
Item	**Comp**	**Description**
BC3S	1,2,3,4,5,6	First set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC4S	1,2,3,4,5,6	Second set of six components of BCC slip on 112 plane. Available for 3D elements only.
BC5S	1,2,3,4,5,6	First set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC6S	1,2,3,4,5,6	Second set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC7S	1,2,3,4,5,6	Third set of six components of BCC slip on 123 plane. Available for 3D elements only.
BC8S	1,2,3,4,5,6	Fourth set of six components of BCC slip on 123 plane. Available for 3D elements only.
FC1H	1,2,3,4,5,6	First set of six components of FCC crystal hardness. Available for 3D elements only.
FC2H	1,2,3,4,5,6	Second set of six components of FCC crystal hardness. Available for 3D elements only.
HC1H	1,2,3,4,5,6	Sixcomponents of HCP crystal hardness on basal and prismatic systems. Available for 3D elements.
HC2H	1,2,3,4,5,6	Six components of HCP crystal hardness on pyramidal system. Available for 3D elements only.
HC3H	1,2,3,4,5,6	First set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC4H	1,2,3,4,5,6	Second set of six components of HCP crystal hardness on the first-order pyramidal system. Available for 3D elements only.
HC5H	1,2,3,4,5,6	Six components of HCP crystal hardness on the second-order pyramidal system. Available for 3D elements only.
BC1H	1,2,3,4,5,6	First set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC2H	1,2,3,4,5,6	Second set of six components of BCC hardness on 111 plane. Available for 3D elements only.
BC3H	1,2,3,4,5,6	First set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC4H	1,2,3,4,5,6	Second set of six components of BCC hardness on 112 plane. Available for 3D elements only.
BC5H	1,2,3,4,5,6	First set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC6H	1,2,3,4,5,6	Second set of six components of BCC hardness on 123 plane. Available for 3D elements only.
BC7H	1,2,3,4,5,6	Third set of six components of BCC hardness on 123 plane. Available for 3D elements only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Item and Component Labels PRNSOL (p. 1505), *Item*, *Comp*		
Item	**Comp**	**Description**
BC8H	1,2,3,4,5,6	Fourth set of six components of BCC hardness on 123 plane. Available for 3D elements only.
XELG	1,2,3,45,6,EQV	Crystal Lagrangian strain in 11, 22, 33, 12, 23,13 directions and its equivalent. Available for 3D elements only.
SINT	RHO, ETA, SSTR, GRAIN	Sintering relative density, viscosity, sintering stress, and average grain size values.

[a] For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . ., TTOP instead of TEMP.

[b] Not supported by PowerGraphics.

[c] Valid only for SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, and SHELL281.

[d] This item outputs nodal-averaged results if they are available on the results file.

[e] Works only if failure criteria are provided (**FC (p. 732)** and **TB (p. 1899)**).

[f] The results for this postprocessing SEND component are invalid for ELBOW290 if that element is used with viscoelastic or viscohyperelastic materials.

[g] USR1 through USR9 require a failure-criteria routine.

[h] Not supported by PowerGraphics.

[i] For contact elements CONTA172, CONTA174, CONTA175, and CONTA177, the reported data are averaged across the element. To obtain a more meaningful STAT value, use **PRESOL (p. 1481)**.

[j] No vector sum is calculated for coupled pore-pressure-thermal (CPT*nnn*) elements.

Table 222: PRNSOL - Selected Result (SRES) Component Labels

PRNSOL (p. 1505),SRES,*Comp*	
Comp	**Description**
SVAR*n*	The *n*th state variable.
FLDUF0*n*	The *n*th user-defined field variable.
ItemComp (concatenated *Item* + *Comp* label) from Table 208: OSRESULT - Item and Component Labels (p. 1328)[a]	See the table (p. 1328) for the combined item and component description.

[a] See Example 10: Specifying and Retrieving Selected Result Output (p. 1329).

Menu Paths

Main Menu>General Postproc>List Results>Nodal Solution

Main Menu>General Postproc>List Results>Sorted Listing>Sort Nodes

Utility Menu>List>Results>Nodal Solution

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PROD, *IR*, *IA*, *IB*, *IC*, *Name*, - -, - -, *FACTA*, *FACTB*, *FACTC*

Multiplies variables.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA, IB, IC

Reference numbers of the three variables to be operated on. If only two leave *IC* blank. If only one, leave *IB* blank also.

Name

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields

FACTA, FACTB, FACTC

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Multiplies variables (up to three at once) according to the operation:

IR = (*FACTA* x *IA*) x (*FACTB* x *IB*) x (*FACTC* x *IC*)

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Multiply

PRORB, *WhrlNodKey*

Prints the orbital motion characteristics of a rotating structure

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WhrlNodKey

Flag to print the whirl for each node:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 (ON or YES)

Print the whirl for each node.

0 (OFF or NO)

No printout. This value is the default.

Notes

When a structure is rotating and the Coriolis or gyroscopic effect is taken into account (**CORIOLIS (p. 417)**), nodes lying on the rotation axis generally exhibit an elliptical orbital motion. The **PRORB (p. 1515)** command prints out the orbit characteristics A, B, PSI, PHI, YMAX, ZMAX, and Whirl of each rotating node, where

A is the semi-major axis.

B is the semi-minor axis.

PSI is the angle between local y axis and major axis.

PHI is the angle between initial position (t = 0) and major axis.

YMAX is the maximum displacement along local y axis.

ZMAX is the maximum displacement along local z axis.

Whirl is the direction of an orbital motion (BW for backward whirl and FW for forward whirl).

Angles PSI and PHI are in degrees and within the range of -180 through +180.

To *display* the characteristics of the orbital path traversed by each node, issue the **PLORB (p. 1437)** command.

The **PRORB (p. 1515)** command is valid for line elements (such as BEAM188, BEAM189, PIPE288, and PIPE289). **PRORB (p. 1515)** is not supported for beam elements with the warping degree of freedom activated.

Your model must also involve a rotational velocity (**OMEGA (p. 1324)** or **CMOMEGA (p. 368)**) with Coriolis enabled (**CORIOLIS (p. 417)**).

Because orbit data is written in the database, a **SET (p. 1724)** command must be issued after the **PRORB (p. 1515)** command to ensure proper output for subsequent postprocessing commands.

The coordinate system for displaying nodal results must be global Cartesian (**RSYS (p. 1639)**,*KCN* = 0). **PRORB (p. 1515)** is not supported if nodes are rotated in a cylindrical coordinate system.

Menu Paths

Main Menu>General Postproc>Rotor Dynamics>Plot orbit

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRPATH, *Lab1*, *Lab2*, *Lab3*, *Lab4*, *Lab5*, *Lab6*

Prints path items along a geometry path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab1*, *Lab2*, *Lab3*, . . . , *Lab6

Labels identifying the path items to be printed. Up to six items may be printed at a time. Predefined path geometry items XG, YZ, ZG, and S (**PDEF (p. 1372)**) may also be printed.

Notes

Prints path items with respect to a geometry path (as defined by the **PATH (p. 1354)** and **PPATH (p. 1463)** commands). Path items and their labels must have been defined with the **PDEF (p. 1372)**, **PVECT (p. 1556)**, **PCALC (p. 1362)**, **PDOT (p. 1375)**, **PCROSS (p. 1371)**, or **PRNEAR (p. 1501)** commands. Path items may also be displayed with the **PLPATH (p. 1439)** and **PLPAGM (p. 1438)** commands. See the **PRANGE (p. 1464)** command for range control of the path.

Fore more information, see Mapping Results onto a Path in the *Basic Analysis Guide*.

Menu Paths

Main Menu>General Postproc>List Results>Path Items

Main Menu>General Postproc>Path Operations>Plot Path Item>List Path Items

Utility Menu>List>Results>Path Data

PRRFOR, *Lab*

Prints the constrained node reaction solution. Used with the FORCE (p. 772) command.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are:

Structural force labels: FX, FY or FZ (forces); F (FX, FY and FZ); MX, MY or MZ (moments); M (MX, MY and MZ).

Thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow).

Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid forces); VF (VFX, VFY and VFZ).

Electric force labels: AMPS (current flow); CHRG (charge); CURT (current); VLTG (voltage drop).

Magnetic force labels: FLUX (magnetic flux); CSGZ (magnetic current segment); CURT (current), VLTG (voltage drop).

Diffusion labels: RATE (diffusion flow rate).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

PRRFOR (p. 1517) has the same functionality as the **PRRSOL (p. 1518)** command; use **PRRFOR (p. 1517)** instead of **PRRSOL (p. 1518)** when a **FORCE (p. 772)** command has been issued.

In a non-spectrum analysis, if either contact or pretension elements exist in the model, **PRRFOR (p. 1517)** uses the **PRRSOL (p. 1518)** command internally and the **FORCE (p. 772)** setting is ignored.

Because modal displacements cannot be used to calculate contact element nodal forces, those forces are not included in the spectrum and PSD analyses reaction solution. As a consequence, the **PRRFOR (p. 1517)** command is not supported when constraints on contact element pilot nodes are present.

PRRFOR (p. 1517) is not valid when using the amplitude or phase results set (*KIMG* = AMPL or PHAS on the **SET (p. 1724)** command). Use **PRRSOL (p. 1518)** instead.

Using PRRFOR (p. 1517) in a Spectrum or PSD Analysis (ANTYPE (p. 162),SPECTR)

When using **PRRFOR (p. 1517)** in a spectrum analysis (after the combination file has been input through **/INPUT (p. 948)**,,MCOM and when **SPOPT (p. 1837)** has not been issued with *Elcalc* = YES during the spectrum analysis), or in a PSD analysis when postprocessing 1-sigma results (loadstep 3, 4, or 5), the following message will display in the printout header:

```
(Spectrum analysis summation is used)
```

This message means that the summation of the element nodal forces is performed prior to the combination of those forces. In this case, **RSYS (p. 1639)** does not apply, and the reaction forces are in the nodal coordinate systems. Unlike **PRRSOL (p. 1518)**, which retrieves the forces from the database, the **PRRFOR (p. 1517)** command calculates the forces in the postprocessor.

The spectrum analysis summation is available when the element results are written to the mode file, `Jobname.mode` (*MSUPkey* = Yes on **MXPAND (p. 1203)**).

The spectrum analysis summation is not available after reading a load case (**LCWRITE (p. 1031)**, **LCZERO (p. 1032)**, **LCASE (p. 1019)**).

Menu Paths

This command cannot be accessed from a menu.

PRRSOL, *Lab*

Prints the constrained node reaction solution.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Nodal reaction load type. If blank, use the first ten of all available labels. Valid labels are:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Structural force labels: FX, FY or FZ (forces); F (FX, FY and FZ); MX, MY or MZ (moments); M (MX, MY and MZ); BMOM (bimoments).

Thermal force labels: HEAT, HBOT, HE2, HE3, . . ., HTOP (heat flow).

Fluid force labels: FLOW (fluid flow); VFX, VFY and VFZ (fluid forces); VF (VFX, VFY and VFZ).

Electric force labels: AMPS (current flow); CHRG (charge); CURT (current); VLTG (voltage drop).

Magnetic force labels: FLUX (magnetic flux); CSGZ (magnetic current segment); CURT (current), VLTG (voltage drop).

Diffusion labels: RATE (diffusion flow rate).

Notes

Prints the constrained node reaction solution for the selected nodes in the sorted sequence. For coupled nodes and nodes in constraint equations, the sum of all reactions in the coupled or constraint equation set appears at the primary node of the set. Results are in the global Cartesian coordinate directions unless transformed (**RSYS (p. 1639)**).

PRRSOL (p. 1518) is not valid if any load is applied to a constrained node in the direction of the constraint and any of the following is true:

- **LCOPER (p. 1026)** has been used.
- **LCASE (p. 1019)** has been used to read from a load case file.
- The applied loads and constraints in the database are not the ones used to create the results data being processed.

PRRSOL (p. 1518) provides the total reaction solution (static, plus damping, plus inertial, as appropriate based on the analysis type); however, modal reactions include only the static contribution.

Use **PRRFOR (p. 1517)** instead of **PRRSOL (p. 1518)** with the **FORCE (p. 772)** command to obtain only the static, damping, or inertial components.

When using distributed-memory parallel processing, in a spectrum analysis or a PSD analysis performed with *Elcalc* = YES on the **SPOPT (p. 1837)** command, use **PRRFOR (p. 1517)** instead of **PRRSOL (p. 1518)** to print the maximum reaction forces (spectrum analysis) or reaction forces variances of 1-σ solutions, as **PRRSOL (p. 1518)** may lead to more conservative results.

Menu Paths

Main Menu>General Postproc>List Results>Reaction Solu

Utility Menu>List>Results>Reaction Solution

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRSCONTROL, *Key*

Specifies whether to include pressure load stiffness in the element stiffness formation.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Pressure load stiffness key. In general, use the default setting. Use a non-default setting only if you encounter convergence difficulties. Pressure load stiffness is automatically included when using eigenvalue buckling analyses (**ANTYPE (p. 162)**,BUCKLE), equivalent to *Key* = INCP. For all other types of analyses, valid arguments for *Key* are:

NOPL

Pressure load stiffness not included for any elements.

(blank) (default)

Include pressure load stiffness for elements SURF153, SURF154, SURF156, SURF159, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, BEAM188, BEAM189, FOLLW201, SHELL208, SHELL209, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.

Notes

This command is rarely needed. The default settings are recommended for most analyses.

Menu Paths

This command cannot be accessed from a menu.

PRSECT, *RH0*, *KBR*, *KBR3D*

Calculates and prints linearized stresses along a section path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RH0

In-plane (X-Y) average radius of curvature of the inside and outside surfaces of an axisymmetric section. If zero (or blank), a plane or 3D structure is assumed. If nonzero, an axisymmetric structure is assumed. Use any large number (or -1) for an axisymmetric straight section.

KBR

Through-thickness bending stresses key for an axisymmetric analysis (*RH0* $\neq$ 0):

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Include the thickness-direction bending stresses.

1

Ignore the thickness-direction bending stresses.

2

Include the thickness-direction bending stress using the same formula as the Y (axial direction) bending stress. Also use the same formula for the shear stress.

KBR3D

Through-thickness bending stresses key for 3D geometry (*RH0* = 0):

0

Include the thickness-direction bending stresses.

1

Ignore the following thickness-direction bending stresses: SX, SXY, SXZ

Notes

You may want to linearize the stresses through a section and separate them into categories for various code calculations. **PRSECT (p. 1520)** calculates and reports linearized stresses along a section path. The linearized stresses are also separated into membrane, bending, membrane plus bending, peak, and total stress categories.

Define your section path (**PATH (p. 1354)** and **PPATH (p. 1463)** with the NODE option). Your path must lie entirely within the selected set of elements (that is, no element gaps may exist along the path). **PATH (p. 1354)** and **PPATH (p. 1463)** only retrieve the two end nodes; the path data is not retained. The section path is defined by the two end nodes, and by 47 intermediate points that are automatically determined by linear interpolation in the active display coordinate system (**DSYS (p. 582)**). The number and location of the intermediate points are not affected by the number of divisions set by **PATH (p. 1354)**,,,,*nDiv*.

Your linearized component stress values are obtained by interpolating each element's average corner nodal values along the section path points within each path element. **PRSECT (p. 1520)** reports the linearized component and principal stresses for each stress category at the beginning, mid-length, and end of the section path. **PRPATH (p. 1517)** can be used to report the total stresses at the intermediate points.

Section paths can be through any set of solid (2D plane, 2D axisymmetric or 3D) elements; however, section paths are usually defined to be through the thickness of the structure and normal to the inner and outer structure surfaces. Section paths (in-plane only) can also be defined for shell element structures.

If the *RH0* option is set to indicate the axisymmetric option (non-zero), **PRSECT (p. 1520)** reports the linearized stresses in the section coordinates (SX – along the path, SY – normal to the path, and SZ – hoop direction). If the *RH0* option is set to indicate the 2D planar or 3D option (zero or blank), **PRSECT (p. 1520)**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reports the linearized stresses in the active results coordinate system (**RSYS (p. 1639)**]. If the *RH0* option is zero or blank and either **RSYS (p. 1639)**, SOLU or **RSYS (p. 1639)**, -1 are active, the linearized stresses are calculated and reported in the global Cartesian coordinate system.

Linearized stress calculations should be performed in a rectangular coordinate system. Principal stresses are recalculated from the component stresses and are invariant with the coordinate system as long as SX is in the same direction at all points along the defined path. The **PLSECT (p. 1440)** command displays the linearized stresses in the same coordinate system as reported by **PRSECT (p. 1520)**.

Stress components through the section are linearized by a line integral method and separated into constant membrane stresses, bending stresses varying linearly between end points, and peak stresses (defined as the difference between the actual (total) stress and the membrane plus bending combination).

For nonaxisymmetric structures, the bending stresses are calculated such that the neutral axis is at the midpoint of the path. Axisymmetric results include the effects of both the radius of revolution (automatically determined from the node locations) and the in-plane average radius of curvature of the section surfaces (user input).

For axisymmetric cases, Mechanical APDL calculates the linearized bending stress in the through-thickness direction as the difference between the total outer fiber stress and the membrane stress if *KBR* = 0. The calculation method may be conservative for locations with a highly nonlinear variation of stress in the through-thickness direction. Alternatively, you can specify *KBR* = 2 to calculate the bending stress using the same method and formula as the Y (axial direction) bending stress. For more information, see the discussion of axisymmetric cases (specifically Equation 17.38) in the *Mechanical APDL Theory Reference*.

In analyses of 3D models with *RH0* = 0, ignoring the calculated out-of-plane bending stresses is recommended in some scenarios when determining the linearized bending stresses. If *KBR3D* = 0, all calculated stresses are included in the linearized bending-stress calculations. If *KBR3D* = 1, these calculated out-of-plane bending stresses are ignored in the linearized bending-stress calculations: SX, SXY, SXZ. (The principal bending-stress calculation for S1, S2, S3, SINT, and SEQV is performed with these zeroed components.)

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER].

Menu Paths

Main Menu>General Postproc>List Results>Linearized Strs

Main Menu>General Postproc>Path Operations>List Linearized

Utility Menu>List>Results>Linearized Stresses

PRTIME, *TMIN*, *TMAX*

Defines the time range for which data are to be listed.

POST26 (p. 58): Listing (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TMIN

Minimum time (defaults to the first point stored).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TMAX

Maximum time (defaults to the last point stored).

Command Default

Use the previously defined range (**TIMERANGE (p. 2001)**).

Notes

Defines the time (or frequency) range (within the range stored) for which data are to be listed.

Menu Paths

Main Menu>TimeHist Postpro>Settings>List

PRVAR, *NVAR1*, *NVAR2*, *NVAR3*, *NVAR4*, *NVAR5*, *NVAR6*

Lists variables vs. time (or frequency).

POST26 (p. 58): Listing (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR1, NVAR2, NVAR3, . . . , NVAR6

Variables to be displayed, defined either by the reference number or a unique thirty-two character name. If duplicate names are used the command will print the data for the lowest-numbered variable with that name.

Notes

Lists variables vs. time (or frequency). Up to six variables may be listed across the line. Time column output format can be changed using the **/FORMAT (p. 774)** command arguments *Ftype*, *NWIDTH*, and *DSIGNF*.

Menu Paths

Main Menu>TimeHist Postpro>List Variables

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRVECT,*Item*,*Lab2*,*Lab3*,*LabP*

Prints results as vector magnitude and direction cosines.

POST1 (p. 51): Results (p. 51)
POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item

Predefined vector item (from Table 223: PRVECT - Valid Item and Component Labels (p. 1524) below) or a label identifying the i-component of a user-defined vector.

Lab2

Label identifying the j-component of a user-defined vector. In most cases, this value must be blank if *Item* is selected from Table 223: PRVECT - Valid Item and Component Labels (p. 1524). Individual principal stresses (*Item* = S) or principal strains (*Item* = EP*xx*) may be printed by specifying the value as 1, 2, or 3.

Lab3

Label identifying the k-component of a user-defined vector. Must be blank if *Item* is selected from list below or for 2D user defined vector.

LabP

Label assigned to resultant vector for printout labeling (defaults to *Item*).

Notes

Prints various solution results as vector magnitude and direction cosines for the selected nodes and/or elements. For example, **PRVECT (p. 1524)**,U prints the displacement magnitude and its direction cosines for all selected nodes. For nodal degree of freedom vector results, direction cosines are with respect to the results coordinate system RSYS. For element results, direction cosines are with respect to the global Cartesian system. Item components may be printed with the **PRNSOL (p. 1505)** command. Various results also depend upon the recalculation method and the selected results location (**LAYER (p. 1015)**, **SHELL (p. 1784)**, **NSEL (p. 1266)**, and **ESEL (p. 661)**). Items may be selected from a set of recognized vector labels (*Item*) or a vector may be defined from up to three scalar labels (*Item*,*Lab2*,*Lab3*). Scalar labels may be user-defined with the **ETABLE (p. 687)** command.

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

Table 223: PRVECT - Valid Item and Component Labels

Item	Comp	Description
Valid item labels for nodal degree of freedom vector results are:		
U		Structural displacement vector magnitude and direction cosines.
ROT		Structural rotation vector magnitude and direction cosines.
V		Velocity vector magnitude and direction cosines.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description
A		Magnetic vector potential vector magnitude and direction cosines.
	Valid item labels for element results are:	
S		Principal stresses and direction cosines.
EPTO		Principal total strains (EPEL + EPPL + EPCR) and direction cosines.
EPEL		Principal elastic strains and direction cosines.
EPPL		Principal plastic strains and direction cosines.
EPCR		Principal creep strains and direction cosines.
EPTH		Principal thermal strains and direction cosines.
EPDI		Principal diffusion strains and direction cosines.
TG		Thermal gradient vector sum and direction cosines.
TF		Thermal flux vector sum and direction cosines.
PG		Velocity or energy density flux (room acoustics) vector sum and direction cosines.
EF		Electric field vector sum and direction cosines.
D		Electric flux density vector sum and direction cosines.
H		Magnetic field intensity vector sum and direction cosines. If *Lab2* is blank, Item is interpreted as one of the predefined labels; otherwise, Item is interpreted as a user-defined **ET (p. 686)** label and the program requests a non-blank *Lab2*/*Lab3* according to the dimension of the problem.
B		Magnetic flux density vector sum and direction cosines.
CG		Concentration gradient vector sum and direction cosines.
DF		Diffusion flux density vector sum and direction cosines.
FMAG		Electromagnetic force vector sum and direction cosines.
P		Poynting vector sum and direction cosines.
JS		Source current density vector sum and direction cosines for low-frequency magnetic analyses. Total current density vector sum and direction cosines (sum of conduction and displacement current densities) in low frequency electric analyses.
JT		Total measurable current density vector sum and direction cosines in low-frequency electromagnetic analyses. (Conduction current density vector sum and direction cosines in a low-frequency electric analysis.)
JC		Conduction current density vector sum and direction cosines for elements that support conduction current calculation.
SNDI		Sound intensity vector sum and direction cosines.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>List Results>Vector Data

Utility Menu>List>Results>Vector Data

PSCONTROL, *Option*, *Key*

Enables or disables shared-memory parallel operations.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Specify the operations for which you intend to enable/disable parallel behavior:

ALL

Enable/disable parallel for all areas (default).

PREP

Enable/disable parallel during preprocessing (**/PREP7 (p. 1479)**).

SOLU

Enable/disable parallel during solution (**/SOLU (p. 1821)**).

FORM

Enable/disable parallel during element matrix generation.

SOLV

Enable/disable parallel during equation solver.

RESU

Enable/disable parallel during element results calculation.

POST

Enable/disable parallel during postprocessing (**/POST1 (p. 1461)** and **/POST26 (p. 1462)**).

STAT

List parallel operations that are enabled/disabled.

Key

Option control key. Used for all *Option* values except STAT.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON

Enable parallel operation.

OFF

Disable parallel operation.

Command Default

None. The command is ignored if issued with no arguments.

Notes

Use this command in shared-memory parallel operations.

This command is useful when you encounter minor discrepancies in a nonlinear solution when using different numbers of processors. A parallel operation applied to the element matrix generation can produce a different nonlinear solution with a different number of processors. Although the nonlinear solution converges to the same nonlinear tolerance, the minor discrepancy created may not be desirable for consistency.

Enabling/disabling parallel behavior for the solution (*Option* = SOLU) supersedes the activation/deactivation of parallel behavior for element matrix generation (FORM), equation solver (SOLV), and element results calculation (RESU).

The SOLV option supports only the sparse direct and PCG solvers (**EQSLV (p. 648)**,SPARSE *or* PCG). No other solvers are supported.

This command applies only to shared-memory architecture. It does not apply to distributed-memory parallel processing.

Menu Paths

This command cannot be accessed from a menu.

PSDCOM,*SIGNIF*,*COMODE*,,*ForceType*

Specifies the power spectral density mode combination method.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIGNIF

Combine only those modes whose significance level exceeds the*SIGNIF* threshold. For PSD response (**SPOPT (p. 1837)**,PSD), the significance level is defined as the modal covariance matrix term, divided by the maximum modal covariance matrix term. Any term whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of terms used. *SIGNIF* defaults to 0.0001. If *SIGNIF* is specified as 0.0, it is taken as 0.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMODE

First *COMODE* number of modes to be actually combined. *COMODE* must always be less than or equal to *NMODE* (input quantity *NMODE* on the **SPOPT (p. 1837)** command). *COMODE* defaults to *NMODE*. *COMODE* performs a second level of control for the first sequential *COMODE* number of modes to be combined. It uses the significance level threshold indicated by *SIGNIF* and operates only on the significant modes.

ForceType

Label identifying the forces to be combined:

STATIC

Combine the modal static forces (default).

TOTAL

Combine the modal static plus inertial forces.

Notes

This command is also valid for PREP7. This command is valid only for **SPOPT (p. 1837)**,PSD.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Mode Combine

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Mode Combine

PSDFRQ, *TBLNO1*, *TBLNO2*, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*

Defines the frequency points for the input spectrum tables PSDVAL vs. PSDFRQ for PSD analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO1

Input table number. When used with the **COVAL (p. 420)** or the **QDVAL (p. 1559)** command, *TBLNO1* represents the row number of this table. Up to 200 tables may be defined.

TBLNO2

Input table number. *TBLNO2* is used only for the **COVAL (p. 420)** or the **QDVAL (p. 1559)** commands and represents the column number of this table.

FREQ1, FREQ2, FREQ3, . . . , FREQ7

Frequency points (cycles/time) for spectrum vs. frequency tables. *FREQ1* should be greater than zero, and values must be in ascending order. Log-log interpolation will be used between frequency points.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The spectrum values may be input with the **PSDVAL (p. 1534)**, **COVAL (p. 420)**, or **QDVAL (p. 1559)** commands. A separate **PSDFRQ (p. 1528)** command must be used for each table and cross table defined. Frequencies must be in ascending order.

Repeat **PSDFRQ (p. 1528)** command for additional frequency points. Values are added after the last nonzero frequency. If all fields after **PSDFRQ (p. 1528)** are blank, all input vs. frequency tables are erased. If *TBLNO1* is nonblank, all corresponding **PSDVAL (p. 1534)** tables are erased. If both *TBLNO1* and *TBLNO2* are nonblank, all corresponding **COVAL (p. 420)** and **QDVAL (p. 1559)** tables are erased.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Cospectral

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Erase Co & Quad

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Erase Tables

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>PSD vs Freq

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Cospectral

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Erase Co & Quad

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Erase Tables

Main Menu>Solution>Load Step Opts>Spectrum>PSD>PSD vs Freq

PSDGRAPH, *TBLNO1*, *TBLNO2*, *DisplayKey*

Displays input PSD curves.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO1

PSD table number to display.

TBLNO2

Second PSD table number to display. *TBLNO2* is used only in conjunction with the **COVAL (p. 420)** or the **QDVAL (p. 1559)** commands.

DisplayKey

Key to display the points markers and numbering:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Display points markers and numbering (default).

1

Display points numbering only.

2

Display points markers only.

3

No points markers or numbering.

Notes

The input PSD tables are displayed in log-log format as dotted lines. The best-fit curves, used to perform the closed-form integration, are displayed as solid lines. If there is a significant discrepancy between the two, then you should add one or more intermediate points to the table to obtain a better fit.

If *TBLN02* is zero, blank, or equal to *TBLN01*, then the autospectra (**PSDVAL (p. 1534)**) are displayed for *TBLN01*. If *TBLN02* is also specified, then the autospectra for *TBLN01* and *TBLN02* are displayed, along with the corresponding cospectra (**COVAL (p. 420)**) and quadspectra (**QDVAL (p. 1559)**), if they are defined.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Loads>Spectrum>Graph PSD Tab

Main Menu>Solution>Spectrum>Graph PSD Tab

PSDRES, *Lab*, *RelKey*

Controls solution output written to the results file from a PSD analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Label identifying the solution output:

DISP

Displacement solution (default). One-sigma displacements, stresses, forces, etc. Written as load step 3 on `File.rst`.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VELO

Velocity solution. One-sigma velocities, "stress velocities," "force velocities," etc. Written as load step 4 of `File.rst`.

ACEL

Acceleration solution. One-sigma accelerations, "stress accelerations," "force accelerations," etc. Written as load step 5 on `File.rst`.

RelKey

Key defining relative or absolute calculations:

REL

Calculations are relative to the base excitation (default).

ABS

Calculations are absolute.

OFF

No calculation of solution output identified by *Lab*.

Command Default

Relative displacement solution, no velocity or acceleration solution for 1 σ results.

Notes

Controls the amount and form of solution output written to the results file from a PSD analysis. One-sigma values of the relative or absolute displacement solution, relative or absolute velocity solution, relative or absolute acceleration solution, or any combination may be included on the results file.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Calc Controls

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Calc Controls

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PSDSPL, *TBLNO*, *RMIN*, *RMAX*

Defines a partially correlated excitation in a PSD analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Input PSD table number defined with **PSDVAL (p. 1534)** command.

RMIN

Minimum distance between excitation points which are partially correlated. Excited nodes closer than *RMIN* will be fully correlated.

RMAX

Maximum distance between excitation points which are partially correlated. Excited nodes farther apart than *RMAX* will be uncorrelated.

Notes

Defines a partially correlated excitation in terms of a sphere of influence relating excitation point geometry (in a PSD analysis). If the distance between any two excitation points is less than *RMIN*, then the excitation is fully correlated. If the distance is greater than *RMAX*, then the excitation is uncorrelated. If the distance lies between *RMIN* and *RMAX*, then the excitation is partially correlated with the degree of correlation dependent on the separation distance between the points. This command is not available for a pressure PSD analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Spatial Correlat

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Spatial Correlat

PSDUNIT, *TBLNO*, *Type*, *GVALUE*

Defines the type of input PSD.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Input table number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type

Label identifying the type of spectrum:

DISP

Displacement spectrum (in terms of displacement^2/Hz).

VELO

Velocity spectrum (in terms of velocity^2/Hz).

ACEL

Acceleration spectrum (in terms of acceleration^2/Hz).

ACCG

Acceleration spectrum (in terms of g^2/Hz).

FORC

Force spectrum (in terms of force^2/Hz).

PRES

Pressure spectrum (in terms of pressure^2/Hz).

GVALUE

Value of acceleration due to gravity in any arbitrary units for Type=ACCG. Default is 386.4 in/sec^2.

Command Default

Acceleration (ACEL) spectrum (acceleration^2/Hz).

Notes

Defines the type of PSD defined by the **PSDVAL (p. 1534)**, **COVAL (p. 420)**, and **QDVAL (p. 1559)** commands.

Force (FORC) and pressure (PRES) type spectra can be used only as a nodal excitation.

GVALUE is valid only when type ACCG is specified. A zero or negative value cannot be used. A parameter substitution can also be performed.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Settings

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PSDVAL, *TBLN0*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

Defines PSD values.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLN0

Input table number being defined.

SV1, SV2, SV3, . . . , SV7

Spectral values corresponding to the frequency points (**PSDFRQ (p. 1528)**). Values are interpreted as defined with the **PSDUNIT (p. 1532)** command.

Notes

Defines PSD values to be associated with the previously defined frequency points.

Repeat **PSDVAL (p. 1534)** command for additional values, up to the number of frequency points (**PSDFRQ (p. 1528)**). Values are added after the last nonzero value.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>PSD vs Freq

Main Menu>Solution>Load Step Opts>Spectrum>PSD>PSD vs Freq

PSDWAV, *TBLN0*, *VX*, *VY*, *VZ*

Defines a wave propagation excitation in a PSD analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLN0

Input PSD table number defined with **PSDVAL (p. 1534)** command.

VX

Global Cartesian X-velocity of traveling wave.

VY

Global Cartesian Y-velocity of traveling wave.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VZ

Global Cartesian Z-velocity of traveling wave.

Notes

Defines a traveling wave in a PSD analysis. This command is not available for a pressure PSD analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Traveling Wave
Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Traveling Wave

---

/PSEARCH, *Pname*

Specifies a directory to be searched for "unknown command" macro files.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Pname

Path name (64 characters maximum, and must include the final delimiter) of the middle directory to be searched. Defaults to the user home directory. If *Pname* = OFF, search only the program and current working directories. If *Pname* = STAT, list the current middle directory and show the **ANSYS_MACROLIB** setting.

Command Default

The middle directory searched is the user home directory.

Notes

Specifies the pathname of a directory for file searches when reading unknown-command macro files.

The search for the files is typically from the program directory, then from the user home directory, and then from the current working directory. The command allows the middle directory searched to be other than the user home directory.

This command is valid only at the Begin level.

Menu Paths

Utility Menu>Macro>Macro Search Path

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PSEL, *Type*, *Pname1*, *Pname2*, *Pname3*, *Pname4*, *Pname5*, *Pname6*, *Pname7*, *Pname8*, *Pname9*, *Pname10*

Selects a path or paths.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Select a new path.

R

Reselect a path from the current set of paths.

A

Additionally select a path and extend the current set of paths.

U

Unselect a path from the current set of paths.

ALL

Restore the full set of paths.

NONE

Unselect the full set of paths.

INV

Invert the current set of paths (selected becomes unselected and vice versa).

Pname1, Pname2, Pname3, . . . , Pname10

Name of existing path(s).

Notes

Selects a path or multiple paths, up to ten. Data are flagged as selected and unselected; no data are actually deleted from the database. There is no default for this command; you must specify a type and pathname.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>Path Operations>Archive Path>Store>Paths in file

Main Menu>Preprocessor>Path Operations>Store>Paths in file

/**PSF**,*Item*,*Comp*,*KEY*,*KSHELL*,*Color*

Shows surface load symbols on model displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Item*,*Comp

Labels identifying the surface load to be shown; see Table 224: /PSF - Valid Item and Component Labels (p. 1539).

KEY

Key to turn surface load symbols on or off:

0

Off (default).

1

On, shown as face outlines. Line surface loads (**SFL (p. 1774)**) on solid model plots are shown as arrows.

2

On, shown as arrows.

3

On, shown as color filled surfaces. Line and area surface loads (**SFL (p. 1774)** and **SFA (p. 1739)**) on solid model plots are shown as arrows.

KSHELL

Visibility key for shell elements.

0

Off (default), surface load symbols are displayed only on visible load faces.

1

On, surface load symbols are displayed even if load face is not visible.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color

Visibility key for contour legend.

ON

The symbols (arrows or face outlines) will show up in color with the legend showing the corresponding color labels (default).

OFF

The contour legend will not be displayed. The symbols (arrows or face outlines) will show up in grey. The size of the arrows will be proportional to the applied load.

Command Default

No surface load symbols are displayed.

Notes

/PSF (p. 1537) determines whether and how to show surface loads on subsequent model displays.

If surface loads are applied to solid model entities, only solid model plots show the load symbols; node and element plots do not show them unless the loads are transferred (**SFTRAN (p. 1782)** or **SBCTRAN (p. 1647)**). Similarly, solid model plots do not show the load symbols if surface loads are applied to nodes and elements. For node and element plots of shell element models, the surface load symbols are shown only if the load face is visible from the current viewing direction.

The effects of the **/PSF** command are not cumulative (that is, the command does not modify an existing setting from a previously issued **/PSF (p. 1537)** command). Only the setting specified via the most recent **/PSF (p. 1537)** command applies.

If you issue a postprocessing (POST1) (p. 51) plot command that produces result contours (such as **PLNSOL (p. 1425)**), **/PSF (p. 1537)** has no effect. This behavior prevents conflicting contours in the graphics window.

When using the radiosity method (*Item* = RDSF and *Comp* = ENCL) with *Key* = 2, the radiation arrows point outward from any element face.

/PSF (p. 1537),STAT displays current **/PSF (p. 1537)** settings, and **/PSF (p. 1537)**,DEFA resets them back to default.

Other useful commands are **/PNUM (p. 1457)**,SVAL,1 to show the values of the surface loads, **/VSCALE (p. 2160)** to change arrow lengths, and **/PBC (p. 1356)** and **/PBF (p. 1360)** to activate other load symbols.

For beam elements, only the colors representing shear (GREEN) and normal (RED) pressures are displayed for the arrows. The color of these arrows does not correspond to the magnitudes in the contour legend. The length of these arrows does, however, correlate to the relative magnitude of the pressures.

For elements SURF159, SOLID272, SOLID273, PIPE288 and PIPE289, **/PSF (p. 1537)** is not available when displaying elements with shapes determined from the real constants or section definition (**/ESHAPE (p. 665)**). For PIPE288 and PIPE289, only external loads applied via **SFBEAM (p. 1746)** are displayed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is valid in any processor.

Table 224: /PSF - Valid Item and Component Labels

Item	Comp	Description	Comments
PRES [1]	NORM (or blank)	Applied pressure normal to the face (real component only).	For element types *other than* SURF153, SURF154 and SURF156.
	NORM	Applied pressure normal to the face (real component).	For element types SURF153, SURF154 and SURF156 with KEYOPT(2) = 0. For supported structural elements (p. 1751) with *KCSYS* = 0. [2]
	TANX	Applied pressure in the element tangential x direction (real component).	
	TANY	Applied pressure in the element tangential-y direction (real component).	
	INRM	Applied pressure normal to the face (imaginary component).	
	ITNX	Applied pressure in the element tangential x direction (imaginary component).	
	ITNY	Applied pressure in the element tangential y direction (imaginary component).	
	LOCX	Applied pressure in the local x direction (real component).	For element types SURF153, SURF154 and SURF156 with KEYOPT(2) = 1. For supported structural elements (p. 1751) with *KCSYS* = 1. [2]
	LOCY	Applied pressure in the local y direction (real component).	
	LOCZ	Applied pressure in the local z direction (real component).	
	ILCX	Applied pressure in the local x direction (imaginary component).	
	ILCY	Applied pressure in the local y direction (imaginary component).	
	ILCZ	Applied pressure in the local z direction (imaginary component).	
	RVEC	Applied pressure in the user-defined direction (real component).	For supported structural elements (p. 1751) with *KCSYS* = 2. [2]
	IVEC	Applied pressure in the user-defined direction (imaginary component).	
CONV	HCOEF	Applied convection (film coefficient).	
	TBULK	Applied convection (bulk temperature).	
RAD	EMIS	Applied radiation (emissivity).	
	TAMB	Applied radiation (ambient temperature).	
RDSF	EMSS	Radiation emissivity.	
	ENCL	Enclosure number.	
FSIN		Fluid-solid interface number.	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Item	Comp	Description	Comments
HFLUX		Applied heat flux.	
FSI		Acoustic fluid-structure interface flag.	
IMPD		Applied acoustic impedance.	
SHLD	COND	Applied conductivity.	
	MUR	Applied relative permeability.	
MXWF		Maxwell force flag.	
INF		Exterior surface flag.	
CHRGS		Applied electric surface charge density.	
BLI		Boundary layer impedance flag.	

1. Pressure loads apply to the element coordinate system (KEYOPT(2) = 0). Adjust appropriately for a local coordinate system (KEYOPT(2) = 1). See Figure : Pressures, Figure : Pressures, and Figure : Pressures in the Element Reference.

2. *KCSYS* is specified when issuing **SFCONTROL (p. 1748)**.

Menu Paths

Utility Menu>PlotCtrls>Symbols

PSMAT,*Fname*,*Ext*,*Matrix*,*Color*

Writes an assembled global matrix to a postscript format that graphically displays nonzero matrix values.

AUX2 (p. 61): Binary File Manipulation (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

This file name is used for the postscript file name. It defaults to the current Jobname if left blank.

Ext

Filename extension (eight-character maximum).

Defaults to `.ps` if left blank.

Matrix

Specify which matrix to write to the output postscript file:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STIFF

Write stiffness matrix to output postscript file. Valid for all types of analyses that write a `.full` file.

MASS

Write mass matrix to output postscript file. Valid for buckling, substructure, and modal analyses. If the `.full` file was generated in a buckling analysis, then this label will write the stress stiffening matrix to the output postscript file.

DAMP

Write damping matrix to output postscript file. Only valid for damped modal analyses.

Color

Specifies whether to display the grid in black and white or in color:

BLACK

Each nonzero coefficient is symbolized by a black square (default).

COLOR

Each nonzero coefficient is symbolized by a colored square. The color depends on the module of the coefficient; the range is from blue for the smallest values to red for the largest values. The color map is:



Notes

This command is used to copy a matrix from the assembled global matrix file (`.full` file) as specified on the **FILE (p. 752)** command and write it in a postscript format to a new file named *Fname.Ext* (defaults to `Jobname.ps`). The matrix is symbolized by a grid in which the black or colored squares represent the nonzero coefficients of the matrix. The `.full` file must be available for this command to work properly.

If the matrix is large, it may be difficult to display the postscript file. In this case, use *Color* = BLACK to reduce the postscript file size.

The assembled global matrix file is created during solution depending on the analysis type, equation solver, and other solution options. By default, the assembled global matrix file is never deleted at the end of solution. For most analysis types, the Sparse direct solver and the ICCG solver write a `.full` file. All mode extraction methods used for buckling and modal analyses write a properly formatted `.full` file to be used with the **PSMAT (p. 1540)** command.

When copying the stiffness matrix for transient and harmonic analyses, be aware that the element mass matrix values (and possibly element damping matrix values) are incorporated into the globally assembled stiffness matrix. Thus, the globally assembled stiffness matrix represents more than the stiffness of the model for these analysis types. Please refer to the *Mechanical APDL Theory Reference* for more details.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **PSMAT (p. 1540)** command is not able to display a lumped mass matrix from a `.full` file generated by a harmonic analysis.

When copying a `.full` file, the rows and columns corresponding to specified constraints (for example, **D (p. 483)** commands) are eliminated from the system of equations and therefore are not written to the `.ps` file. In addition, rows and columns corresponding to eliminated (dependent) degrees of freedom from coupling and/or constraint equations (for example, **CE (p. 297)**, **CP (p. 421)** commands) are eliminated from the system of equations and are not written to the `.ps` file. The DOFs that are eliminated from any coupling and/or constraint equations are determined internally by the solution code and may not match what you specified via the **CE (p. 297)**/**CP (p. 421)** (or similar) commands.

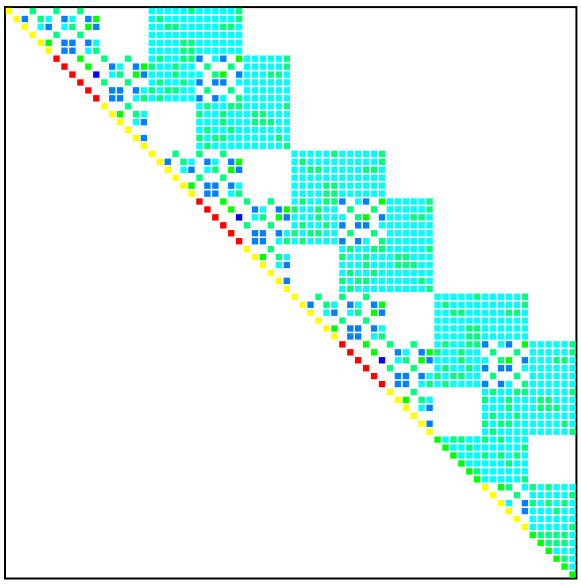
When copying a `.full` file, only the upper triangular part of the matrix will be written to the `.ps` file if the matrix is symmetric; the full matrix is written if the matrix is unsymmetric. Only matrix coefficients that are greater than zero will be written.

The **WRFULL (p. 2184)** command, in conjunction with the **SOLVE (p. 1822)** command, can be used to generate the assembled global matrix file and eliminate the equation solution process and results output process.

The following command sequence shows typical usage of this command.

```
/BATCH,LIST
/AUX2   ! Enter AUX2 processor
FILE,job1,full  ! FULL file containing stiffness matrix is job1.full
PSMAT,job1KColor,ps,STIFF,COLOR ! Create file job1KColor.ps in color
!                               postscript format for stiffness matrix
PSMAT,job1MBlack,,STIFF,BLACK ! Create file job1MBalck.ps in black/white
!                               postscript format for stiffness matrix
FINISH
```

Below is an example of an export of the stiffness matrix to a postscript format using the COLOR option.



K Matrix

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PSMESH,*SECID*,*Name*,*P0*,*Egroup*,*NUM*,*KCN*,*KDIR*,*VALUE*,*NDPLANE*,*PSTOL*,*PSTYPE*,*ECOMP*,*NCOMP*

Creates and meshes a pretension section (PRETS179) or a preload section (MPC184).

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SECID

Unique section number. This number must not already be assigned to a section.

Name

Unique eight character descriptive name, if desired.

P0

Pretension node number (for a pretension section using PRETS179) or joint element number (for a preload section using MPC184).

For a pretension element, the node is defined if it doesn't exist, and the number defaults to the highest node number plus one.

For a joint element, a unique element number is assigned by default.

Egroup*, *NUM

Element group on which **PSMESH (p. 1543)** will operate. If *Egroup* = P, graphical picking is enabled and *NUM* is ignored (valid only in the GUI).

L (or LINE)

PSMESH (p. 1543) operates on all elements in the line specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **LCLEAR (p. 1024)** operation of *NUM* deletes the pretension elements and nodes created by **PSMESH (p. 1543)**. (MPC184 joint elements and associated contact pairs of a preload section are not deleted by **LCLEAR (p. 1024)**.)

A (or AREA)

PSMESH (p. 1543) operates on all elements in the area specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **ACLEAR (p. 93)** of *NUM* deletes the pretension elements and nodes created by **PSMESH (p. 1543)**. (MPC184 joint elements and associated contact pairs of a preload section are not deleted by **ACLEAR (p. 93)**.)

V (or VOLU)

PSMESH (p. 1543) operates on all elements in the volume specified by *NUM*. New pretension nodes are associated with *NUM* or entities below it. Any subsequent **VCLEAR (p. 2076)** of *NUM* deletes the pretension elements and nodes created by **PSMESH (p. 1543)**. (MPC184 joint elements and associated contact pairs of a preload section are not deleted by **VCLEAR (p. 2076)**.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P

PSMESH (p. 1543) operates on elements selected through the subsequent picking operations, and *NUM* is ignored

ALL

The command operates on all selected elements, and *NUM* is ignored.

KCN

Coordinate system number for the separation surface and normal direction.

KDIR

Direction (x, y, or z) normal to separation surface in the *KCN* coordinate system.

If *KCN* is Cartesian, the pretension section normal will be parallel to the *KDIR* axis regardless of the position of the pretension node.

If *KCN* is non-Cartesian, the pretension section normal will be aligned with the *KDIR* direction of system *KCN* at the position of the pretension node.

For an MPC184 joint element defined as part of a preload section, *KDIR* is used to define the normal of the separation surface and does not affect the axis direction of the joint element.

VALUE

Point along the *KDIR* axis at which to locate the separation surface. Ignored if *NDPLANE* is supplied.

NDPLANE

Existing node that **PSMESH (p. 1543)** will use to locate the separation surface. If *NDPLANE* is supplied, the location of the separation surface is defined by the *KDIR* coordinate of *NDPLANE*.

PSTOL

Optional tolerance below *VALUE*. Allows nodes occurring precisely at or slightly below the separation to be identified properly as above the plane. Has the effect of shifting the plane down by *PSTOL*. The following expression represents the default value:

$$\frac{\sqrt{\Delta X^2+\Delta Y^2+\Delta Z^2}}{1000}$$

where ΔX, ΔY, and ΔZ are the dimensions of the locally selected region of the model based on nodal locations (that is, $\Delta X = X_{max} - X_{min}$).

PSTYPE

Type of pretension or preload section to be generated.

If a positive value is specified (or if this argument is left blank), a pretension section that includes PRETS179 elements is generated. The value entered is the element type number for PRETS179. If no number is specified, the program defines the element type number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If TORQUE is specified, a preload section that includes a cylindrical joint element (MPC184) is generated as follows:

- **For a 2D model**: An x-axis cylindrical joint element is generated along with two force-distributed surface-based constraints. A local Cartesian coordinate system is created at the first node of the joint element such that the local x- axis is the axis of that element (KEYOPT(4) = 0 for the MPC184 element).
- **For a 3D model**: A z-axis cylindrical joint element is generated along with two force-distributed surface-based constraints. A local Cartesian coordinate system is created at the first node of the joint element such that the local z- axis is the axis of that element (KEYOPT(4) = 1 for the MPC184 element).
- **For a 3D model that contains beam elements**: A z-axis cylindrical joint element is generated between the endpoints of two beam elements. (No force-distributed surface-based constraints are needed.) A local Cartesian coordinate system is created at the first node of the joint element such that the local z- axis is the axis of that element (KEYOPT(4) = 1 for the MPC184 element).

If a negative value is specified, a preload section that includes a screw joint element (MPC184) is created with the absolute value of *PSTYPE* used as the pitch value for the joint. This option is only valid for 3D models. Two force-distributed surface-based constraints are generated at the cutting surfaces, except for the case of a beam model which does not need the force-distributed constraints. A local Cartesian coordinate system is created at the first node of the joint element such that the local z- axis is the axis of that element.

ECOMP

If specified, the name of a component to be composed of new pretension elements and existing elements modified by the **PSMESH (p. 1543)** command. This argument is not used with the MPC184 joint element.

NCOMP

Name of a component to be composed of nodes on new pretension elements. This argument is not used with the MPC184 joint element.

Notes

PSMESH (p. 1543) generates a pretension section (PRETS179) or a preload section (MPC184) for modeling bolt fastener preloads. The type of section is specified by the *PSTYPE* argument.

The **PSMESH (p. 1543)** command is valid for structural analyses only.

Pretension Section (PRETS179)

When *PSTYPE* is a positive value or blank, the **PSMESH (p. 1543)** command creates a pretension section normal to the pretension load direction by cutting the mesh along existing element boundaries at the point defined by *VALUE* or *NDPLANE* and inserting PRETS179 elements. The **PSMESH (p. 1543)** command verifies that *PSTYPE* is a PRETS179 element type; if it is not, the command finds the lowest available *ITYPE* (**ET (p. 686)**) that is PRETS179, or it creates a new one if necessary.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When it is necessary to define the pretension node, the program uses node *NDPLANE*. If the *NDPLANE* value is not specified, the program defines the pretension node at:

- The centroid of geometric entity *NUM*, if *Egroup* = LINE, AREA, or VOLU; or
- The centroid location of all selected elements, if *Egroup* = ALL or if graphical picking is used.

If the elements to which the pretension load is to be applied have already been meshed in two groups, **PSMESH (p. 1543)** cannot be used to insert the pretension elements. The **EINTF (p. 614)** command must be used to insert the PRETS179 elements between the two meshed groups.

The **PSMESH (p. 1543)** operation copies any nodal temperatures you have defined on the split surface of the original mesh from the original nodes to the newly created coincident duplicate nodes. However, displacements, forces, and other boundary conditions are not copied.

By mathematical definition, the pretension surface must always be a flat plane. In a non-Cartesian coordinate system, the **PSMESH (p. 1543)** command creates that plane at the indicated position, oriented with respect to the specified direction of the active system (in the same manner that the **NROTAT (p. 1264)** command orients a nodal system with respect to a curved system). For example, assuming X = 1 and Y = 45 in a cylindrical coordinate system with Z as the axis of rotation (*KCN* = 1), a pretension surface normal to X tilts 45 degrees away from the global X axis.

A pretension section can be defined for fastener models made up of any 2D or 3D structural solid, beam, shell, pipe, or link element type. The elements can be low- or high-order.

The pretension section is also supported for general axisymmetric elements (SOLID272 and SOLID273). **PSMESH (p. 1543)** cuts the model and generates PRETS179 elements between all Fourier nodes in the circumferential direction.

For more information, see Defining Pretension in a Joint Fastener in the *Basic Analysis Guide*.

Preload Section (MPC184)

When *PSTYPE* is a negative value or set to TORQUE, the **PSMESH (p. 1543)** command defines an MPC184 joint element for applying a preload to a bolt undergoing large rotation or large deformation. **PSMESH (p. 1543)** cuts the mesh in two parts along existing element boundaries at the point defined by *VALUE* or *NDPLANE*. It generates force-distributed surface-based constraints (remote points) on the cutting surfaces, inserts an MPC184 joint element that connects the two pilot nodes, and creates a local Cartesian coordinate system at the first node of the joint element to define the normal direction. If the joint is between beam elements, no force-distributed constraints are generated. For more information, see Defining Preload in a Joint Fastener Undergoing Large Rotation in the *Basic Analysis Guide*.

The preload section based on MPC184 is not supported for general axisymmetric elements (SOLID272 and SOLID273).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Area

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Line

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Elements in Volu

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Picked Elements

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>Selected Element

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Area

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Line

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Volu

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Picked Elements

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Node>Selected Element

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Area

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Line

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Volu

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Picked Elements

Main Menu>Preprocessor>Modeling>Create>Elements>Pretension>Pretensn Mesh>With Options>Divide at Valu>Selected Element

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Area

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Line

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Elements in Volu

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Picked Elements

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>Selected Element

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Area

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Line

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Elements in Volu

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Picked Elements

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Node>Selected Element

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Area

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Line

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Elements in Volu

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Picked Elements

Main Menu>Preprocessor>Sections>Pretension>Pretensn Mesh>With Options>Divide at Valu>Selected Element

---

/**PSPEC**,*PCOLOR*,*KFILL*,*KBORDR*

Creates annotation polygon attributes (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

PCOLOR

Polygon color (0 ≤ *PCOLOR* ≤ 15):

0

Black.

1

Red-Magenta.

2

Magenta.

3

Blue-Magenta.

4

Blue.

5

Cyan-Blue.

6

Cyan.

7

Green-Cyan.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8

Green.

9

Yellow-Green.

10

Yellow.

11

Orange.

12

Red.

13

Dark Gray.

14

Light Gray.

15

White.

KFILL

Polygon fill key:

0

Hollow polygon.

1

Filled polygon.

KBORDR

Polygon border key:

0

No border.

1

Border.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Creates annotation polygon attributes to control certain characteristics of the polygons created via **/POLYGON (p. 1460)**, **/PMORE (p. 1454)**, **/PCIRCLE (p. 1370)**, and **/PWEDGE (p. 1557)**.

This command is generated by the graphical user interface (GUI) and appears in the log file (`Jobname.log`) if annotation is used. It is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

/PSTATUS, *WN*

Displays the global or window display specifications.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number for status (defaults to global specifications).

Notes

Displays the current global or window display specifications. Global display specifications are common to all windows (e.g. **/SHOW (p. 1785)**, etc.). Window display specifications are specific to one window (e.g. **/VIEW (p. 2131)**, **/TYPE (p. 2034)**, etc.).

This command is valid in any processor.

Menu Paths

Utility Menu>List>Status>Graphics>General

Utility Menu>List>Status>Graphics>Window 1

Utility Menu>List>Status>Graphics>Window 2

Utility Menu>List>Status>Graphics>Window 3

Utility Menu>List>Status>Graphics>Window 4

Utility Menu>List>Status>Graphics>Window 5

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PSTRES, *Key*

Specifies whether prestress effects are calculated or included.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Prestress key:

OFF

Do not calculate (or include) prestress effects (default).

ON

Calculate (or include) prestress effects.

Notes

The **PSTRES (p. 1551)** command specifies whether or not prestress effects are to be calculated or included. The command should be issued *after* the **ANTYPE (p. 162)** command.

Prestress effects are calculated in a static or transient analysis for inclusion in a buckling, modal, harmonic (Method = FULL), or substructure generation analysis. If used in the solution processor (**/SOLU (p. 1821)**), this command is valid only within the first load step.

If you apply thermal body forces during a static analysis to calculate prestress effects, do not delete the forces during any subsequent full harmonic analyses. If you delete the thermal body forces, the thermal prestress effects will not be included in the harmonic analysis. Temperature loads used to define the thermal prestress will also be used in the full harmonic analysis as sinusoidally time-varying temperature loads.

A prestress effect applied with non-follower loads resists rigid body rotation of the model. For example, an unsupported beam with axial tensile forces applied to both ends will have two nonzero rotational rigid body modes.

If tabular loading (***DIM (p. 530)**,,TABLE (p. 533)) was used in the prestress static analysis step, the corresponding value of **TIME (p. 2000)** will be used for tabular evaluations in the modal analysis.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/PSYMB, *Label*, *KEY*

Shows various symbols on displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Show symbols as selected from the following labels:

CS

Local coordinate systems.

NDIR

Nodal coordinate systems (on rotated nodes only).

ESYS

Element coordinate systems (element displays only).

LDIR

Line directions (line displays only).

LDIV

Controls the display of element divisions on lines.

ADIR

Area direction symbol (for keypoint, line, area and volume plots).

LAYR

Layer orientations (relative to the projected element x-axis) or fiber orientations in smeared reinforcing elements. Used only within an element display. Use *KEY* to specify the layer number.

ECON

Element mesh symbols on keypoints and lines.

DOT

Larger symbols displayed for node and keypoint locations. When *Label* = DOT, *KEY* = 1 by default.

XNOD

Extra node of surface or circuit elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FBCS

Force boundary condition scaling. Subsequent *KEY* value determines whether or not to scale the applied and derived forces/moments to the same maximum value.

DEFA

Resets the symbol keys so that Mechanical APDL displays none of the symbols controlled by the **/PSYMB (p. 1552)** command. The value of the *KEY* field is ignored.

STAT

Prints the status of the settings of the symbol keys controlled by the **/PSYMB (p. 1552)** command. The *KEY* field is ignored.

MARK

Controls the marker size (**/GMARKER (p. 861)**). When *Label* = MARK, *KEY* = 10 by default.

KEY

Symbol key:

-1

Effective only if *Label* = LAYR and solid shape element display (**/ESHAPE (p. 665)**) is active. Orientation of all layers appears with the solid shape element display.

0

No symbol (default). If *Label* = LDIV, then *KEY*= 0 indicates that the displayed element divisions will correspond to the existing mesh (the word MESHED or EXISTING can also be substituted). Also, for *Label* = LDIV, if you execute any meshing command (such as **AMESH (p. 121)** or **VMESH (p. 2140)**), *KEY* is set to 0 (MESHED) automatically. If *Label* = FBCS, then *KEY*= 0 indicates that boundary condition scaling will not be common. The applied and derived forces/moments will be scaled to their respective maximum values.

1

Include symbol. If *Label* = LDIV, then *KEY* = 1 indicates that the displayed line divisions will correspond to the value assigned by **LESIZE (p. 1041)** (the word LESIZE can also be substituted). Also, for *Label* = LDIV, if you execute the **LESIZE (p. 1041)** command, *KEY* is set to 1 (LESIZE) automatically. If *Label* = FBCS, then *KEY*= 1 indicates that boundary condition scaling will be common. The applied and derived forces/moments will be scaled to the same maximum value.

N

If *Label* = LAYR, then *N* is equal to the layer number. If *Label* = DOT, then *N* can be equal to 0,1,.....15, indicating the dot size. If *Label* = MARK, then *N* can be equal to 1,…..10, indicating the marker size. If *Label* = LDIV, then *KEY* = -1, indicates that no element divisions will be displayed (the word OFF can also be substituted).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Includes various symbols on the display. Triads are right-handed with x displayed as the longest leg. Where color is displayed, x is white, y is green, and z is blue. For beams, x is always along the length of the element. For lines, an arrow represents the direction of a line, from the beginning keypoint to the end keypoint. See **/PLOPTS (p. 1435)** command for additional display options. Use **/PSTATUS (p. 1550)** or **/PSYMB (p. 1552)**,STAT to display settings. Use **/PSYMB (p. 1552)**,DEFA to reset all specifications back to their defaults. The command **/PSYMB (p. 1552)**,ECON,1 causes the symbol "M" to be displayed on keypoints and lines associated with meshed entities. When you issue the command **/PSYMB (p. 1552)**,DOT,1, a larger symbol is displayed for each node and keypoint location. Using **/PSYMB (p. 1552)**,MARK,1, a smaller marker size can be displayed.

PowerGraphics (**/GRAPHICS (p. 866)**,POWER) does not support **/PSYMB (p. 1552)**,ESYS and **/PSYMB (p. 1552)**,LAYR.

If *KEY* = *N* and PowerGraphics is off, the centroid of the surface elements is connected to the extra node using a gray line. However, if PowerGraphics is on, the color of the line connecting the centroid to the extra node is the same as that for the elements themselves (as determined by **/PNUM (p. 1457)**).

When *Label* = LAYR, the layer systems can be visualized with all current-technology layered elements and the smeared reinforcing element REINF265. To use **/PSYMB (p. 1552)**,LAYR with REINF265, first set the vector-mode graphics option (**/DEVICE (p. 523)**,VECTOR,1).

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Areas

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>On Lines

Main Menu>Preprocessor>Modeling>Move / Modify>RotateNode>To Surf Norm>with Area

Utility Menu>PlotCtrls>Symbols

PSYS, *KCN*

Sets the PML element coordinate system attribute pointer.

PREP7 (p. 22): Artificially Matched Layers (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

Coordinate system number:

0

Use the global Cartesian coordinate system as the PML element coordinate system (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N

Set the PML element coordinate system orientation based on a local Cartesian coordinate system N (where N must be greater than 10) defined by the **LOCAL (p. 1059)** or **CS (p. 435)** command (for example: **LOCAL (p. 1059)**,11,0).

Command Default

The PML element coordinate system orientation defaults to the global Cartesian system.

Notes

This command identifies the local coordinate system used to define the PML (perfectly matched layers) coordinate system of subsequently defined PML elements. It is only applicable to volume elements that support PML. The use of PML coordinate systems is similar to element coordinate systems, as discussed in Understanding the Element Coordinate System. N refers to the coordinate system reference number (*KCN*) defined using the **LOCAL (p. 1059)** (or similar) command.

Menu Paths

This command cannot be accessed from a menu.

PTR, *LOC*, *BASE*, *LOCH*, *BASEH*

Dumps the record of a binary file.

AUX2 (p. 61): Binary File Dump (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LOC*, *BASE

Dump the file record starting at pointer *LOC*. *BASE* is the base pointer, and would be used if *LOC* is a relative pointer.

LOCH*, *BASEH

Second 32-bit integer (if required) for defining the 64-bit pointer.

Notes

Dumps the record of the file named on the AUX2 **FILEAUX2 (p. 753)** command according the format specified on the **FORM (p. 773)** command.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PTXY, *X1*, *Y1*, *X2*, *Y2*, *X3*, *Y3*, *X4*, *Y4*

Defines coordinate pairs for use in polygons and prisms.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X1, Y1, X2, Y2, X3, Y3, X4, Y4

X and Y coordinate pairs on the working plane.

Notes

Defines coordinate pairs for use in polygons and prisms (**POLY (p. 1460)**, **RPRISM (p. 1618)**). The coordinates must be in the Cartesian coordinate system. The coordinate pairs must be input in a continuous order. **PTXY (p. 1556)** may be repeated (up to 100 pairs) until the required pairs have been defined. The pairs will be saved until either the **POLY (p. 1460)** or **PRISM (p. 1497)** command is entered. Use **PTXY (p. 1556)**,STAT to list the saved coordinate pairs. Use **PTXY (p. 1556)**,DELE to delete all the saved coordinate pairs. See the **RPOLY (p. 1616)**, **RPRISM (p. 1618)**, and **RPR4 (p. 1617)** commands for other ways to create polygons and prisms.

Menu Paths

This command cannot be accessed from a menu.

PVECT, *Oper*, *LabXR*, *LabYR*, *LabZR*

Interpolates a set of items onto a path.

POST1 (p. 51): Path Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Valid operations for geometry operations along a path are:

NORM

Defines a unit normal vector at each interpolation point in the direction of the cross product of the tangent to the path and the active Z axis. Resulting vector components are in the active coordinate system (which must be Cartesian).

TANG

Defines a unit vector tangent to the path at each interpolation point. Vector components are in the active coordinate system (which must be Cartesian).

RADI

Defines the position vector of each interpolation point of the path from the center of the active coordinate system (which must be Cartesian).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LabXR

Label (8 characters maximum) assigned to X-component of the resulting vector.

LabYR

Label (8 characters maximum) assigned to Y-component of the resulting vector.

LabZR

Label (8 characters maximum) assigned to Z-component of the resulting vector.

Notes

Defines and interpolates a set of labeled path items along predefined path (**PATH (p. 1354)**) and performs various geometric operations on these path items. A path item must be defined before it can be used with other path operations. Additional path items may be defined with the **PDEF (p. 1372)**, **PCALC (p. 1362)**, **PDOT (p. 1375)**, and **PCROSS (p. 1371)** commands. Path items may be listed or displayed with the **PLPATH (p. 1439)**, **PLPAGM (p. 1438)** and **PRPATH (p. 1517)** commands. Path geometry items (XG, YG, ZG, S) are automatically interpolated (in the active CSYS) if not done so previously with the **PDEF (p. 1372)** command.

Menu Paths

Main Menu>General Postproc>Path Operations>Unit Vector

/**PWEDGE**,*XCENTR*,*YCENTR*,*XLRAD*,*ANGLE1*,*ANGLE2*

Creates an annotation wedge (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCENTR

Wedge X center location (-1.0 < X < 2.0).

YCENTR

Wedge Y center location (-1.0 < Y < 1.0).

XLRAD

Wedge radius length.

ANGLE1

Starting angle of wedge.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANGLE2

Ending angle of wedge. The wedge is drawn counterclockwise from the starting angle, *ANGLE1*, to the ending angle, *ANGLE2*.

Notes

Creates an annotation wedge to be written directly onto the display at a specified location. This is a command generated by the Graphical User Interface (GUI) and will appear in the log file (`Jobname.log`) if annotation is used.

The command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

All wedges are shown on subsequent displays unless the annotation is disabled or deleted. Issue **/LSPEC (p. 1081)** and **/PSPEC (p. 1548)** to set the attributes of the wedge.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Q Commands

QDVAL, *TBLN01*, *TBLN02*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

Defines PSD quadspectral values.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLN01

First input PSD table number associated with this spectrum.

TBLN02

Second input PSD table number associated with this spectrum.

SV1, SV2, SV3, . . . , SV7

PSD quadspectral values corresponding to the frequency points (**PSDFRQ (p. 1528)**).

Notes

Defines PSD quadspectral values to be associated with the previously defined frequency points. Repeat **QDVAL (p. 1559)** command with the same table number for additional points. Unlike autospectra (**PSDVAL (p. 1534)**), the quadspectra can be positive or negative. The quadspectral curve segment where there is a sign change is interpolated linearly (the rest of the curve segments use log-log interpolation). For better accuracy, choose as small a curve segment as possible wherever a sign change occurs.

Two table numbers are required since values are off-diagonal terms. This command is valid for **SPOPT (p. 1837)**,PSD only.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Correlation>Quadspectral

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

QRDOPT,*ReuseKey*,--,--,*SymMeth*,*CMCCoutKey*

Specifies additional QRDAMP modal analysis options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1144)

ReuseKey

Reuse key for method=*QRDAMP* specified in **MODOPT (p. 1140)** command.

ON

Reuse the symmetric eigensolution from the previous load steps or from the previous solution.

OFF

Do not reuse (calculates symmetric eigensolution at current load step). This is the default.

--,--

Unused fields.

SymMeth

Mode-extraction method to be used for the symmetric eigenvalue problem.

LANB

Block Lanczos algorithm (default).

SUBSP

Subspace algorithm.

SNODE

Supernode algorithm.

CMCCoutKey

Complex Modal Contribution Coefficients (CMCC) output key. See Calculate the Complex Mode Contribution Coefficients (CMCC) in the *Structural Analysis Guide* for details and usage.

ON

Output the CMCC to the text file `Jobname.CMCC`.

OFF

Do not output the CMCC. This is the default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

If the `filename.modesym` file exists in the working directory and *ReuseKey* = ON, `filename.modesym` will be reused. If `filename.modesym` does not exist in the working directory, the symmetric eigensolution will be calculated.

When *ReuseKey*=ON, both the new modal analysis (`filename.modesym` usage) and the preceding modal analysis (`filename.modesym` generation) must be performed using the same product version number.

Menu Paths

This command cannot be accessed from a menu.

QSOPT, *Opt*

Specifies quasi static radiation options.

SOLUTION (p. 38): Radiosity (p. 50)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Opt

Quasi static option:

OFF

Do not run transient radiation problem to steady-state (default).

ON

Run transient radiation problem to steady-state.

Notes

For more information on solving a static problem using a false transient approach, see Further Options for Static Analysis in the *Thermal Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Quasi-Static

Main Menu>Solution>Load Step Opts>Time/Frequenc>Quasi-Static

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

QUAD,*NODE1*,*NINTR*,*NODE2*,*NFILL*,*NSTRT*,*NINC*,*PKFAC*

Generates a quadratic line of nodes from three nodes.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE1

Begin fill-in from this node location. If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

NINTR

Intermediate or guiding node. Quadratic curve will pass through this location. *NINTR* may have any node number and any location. If the quadratic line also generates a node with number *NINTR*, the generated location overrides the previous *NINTR* location.

NODE2

End quadratic fill-in at this node location.

NFILL

Fill-in *NFILL* nodes between *NODE1* and *NODE2* (defaults to |*NODE2*-*NODE1*|-1). *NFILL* must be positive.

NSTRT

Node number assigned to first filled-in node (defaults to *NODE1* + *NINC*).

NINC

Add this increment to each of the remaining filled-in node numbers (may be positive or negative). Defaults to (*NODE2*-*NODE1*)/(*NFILL* + 1), that is, linear interpolation.

PKFAC

Peak location factor. If *PKFAC*=0.5, the peak of the quadratic shape occurs at the *NINTR* location. If 0.0 < *PKFAC* < 0.5, the peak occurs to the *NODE2* side of the *NINTR* location. If 0.5 < *PKFAC* < 1.0, the peak occurs to the *NODE1* side of the *NINTR* location. Defaults to 0.5.

Notes

Generates a quadratic line of nodes (in the active coordinate system) from three nodes. The three nodes determine the plane of the curve and may have been defined in any coordinate system. Any number of nodes may be filled-in and any node number sequence may be assigned.

The quadratic line feature uses three nodes (*NODE1*,*NINTR*,*NODE2*) to determine the plane of the curve. The curve passes through the three points, beginning from *NODE1*, through the intermediate (or guiding) point *NINTR*, and toward *NODE2*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Generated nodes are also quadratically spaced. If the guiding node number is within the set being generated, it will be relocated according to the quadratic spacing.

The peak location factor is used to determine how the quadratic fits through the three points. Various nodal progressions can be obtained by different combinations of *PKFAC* and the guiding node location. If the guiding node is at mid-length between *NODE1* and *NODE2*, 0.293 ≤ *PKFAC*< 0.707 will ensure that all generated nodes fall within the *NODE1*,*NODE2* bounds. In the limit, as *PKFAC* approaches 0.0, the peak approaches the line through *NODE1* and *NINTR* at an infinite distance from *NODE1*. The **QUAD (p. 1562)** command generates quadratic lines of nodes, which in turn may be used as a base line for generating irregular surfaces of nodes (by repeating (***REPEAT (p. 1586)**), generating (**NGEN (p. 1218)**, **NSCALE (p. 1265)**), etc.). Irregular surfaces may also be generated with the meshing commands.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Nodes>Quadratic Fill

/QUIT

Exits a processor.

SESSION (p. 11): Processor Entry (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is an alternative to the **FINISH (p. 758)** command. If any cleanup or file writing is normally done by the **FINISH (p. 758)** command, it is bypassed if the **/QUIT (p. 1563)** command is used instead. A new processor may be entered after this command. See the **/EXIT (p. 708)** command to terminate the run.

This command is valid in any processor. This command is not valid at the Begin level.

Menu Paths

This command cannot be accessed from a menu.

QUOT, *IR*, *IA*, *IB*, - - , *Name*, - - , - - , *FACTA*, *FACTB*

Divides two variables.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IA, IB

Reference numbers of the two variables to be operated on.

--

Unused field.

Name

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields.

FACTA, FACTB

Scaling factors (positive or negative) applied to the corresponding variables (default to 1.0).

Notes

Divides two variables according to the operation:

$$IR = (FACTA \times IA)/(FACTB \times IB)$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Divide

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

R Commands

R, *NSET*, *R1*, *R2*, *R3*, *R4*, *R5*, *R6*

Defines the element real constants.

PREP7 (p. 22): Real Constants (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET

Real constant set identification number (arbitrary). If same as a previous set number, set is redefined. Set number relates to that defined with the element (**REAL (p. 1579)**). Note that the GUI automatically assigns this value.

R1, R2, R3, . . . , R6

Real constant values (interpreted as area, moment of inertia, thickness, etc., as required for the particular element type using this set), or table names for tabular input of boundary conditions. Use **RMORE (p. 1614)** command if more than six real constants per set are to be input.

Notes

Defines the element real constants. The real constants required for an element are shown in the Input Summary of each element description in the *Element Reference*. Constants must be input in the same order as shown in that table. If more than the required number of element real constants are specified in a set, only those required are used. If fewer than the required number are specified, zero values are assumed for the unspecified constants.

If using table inputs (COMBIN14, FLUID116, SURF151, SURF152, CONTA172, CONTA174, CONTA175, CONTA177, and COMBI214 only), enclose the table name in % signs (for example, `%tabname%`).

When copying real constants to new sets, Ansys, Inc. recommends that you use the command input. If you do use the GUI, restrict the real constant copy to only the first six real constants (real constants seven and greater will be incorrect for both the master and copy set).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Add/Edit/Delete

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Mass

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech

Main Menu>Preprocessor>Modeling>Create>Circuit>Edit Real Cnst

Main Menu>Preprocessor>Real Constants>Add/Edit/Delete

Main Menu>Solution>Load Step Opts>Other>Real Constants>Add/Edit/Delete

---

RACE, *XC*, *YC*, *RAD*, *TCUR*, *DY*, *DZ*, - - , - - , *Cname*

Defines a "racetrack" current source.

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XC

Location of the mid-thickness of the vertical leg along the working plane X-axis.

YC

Location of the mid-thickness of the horizontal leg along the working plane Y-axis.

RAD

Radius of curvature of the mid-thickness of the curves in the racetrack source. Defaults to .501 * DY

TCUR

Total current, amp-turns (MKS), flowing in the source.

DY

In-plane thickness of the racetrack source.

DZ

Out-of-plane thickness (depth) of the racetrack source.

- -, - -

Unused fields

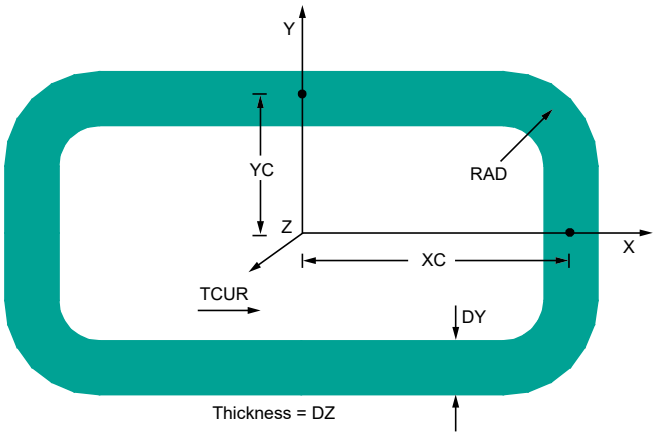
Cname

An alphanumeric name assigned as a component name to the group of SOURC36 elements created by the command macro. Cname must be enclosed in single quotes in the **RACE (p. 1566)** command line. Cname may be up to 32 characters, beginning with a letter and containing only letters, numbers, dots (.), and underscores (_). Component names beginning with an underscore (e.g., _LOOP) are reserved for use by Mechanical APDL and should be avoided. If blank, no component name is assigned.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

RACE (p. 1566) invokes a Mechanical APDL macro which defines a racetrack current source in the working plane coordinate system.



The current source is generated from bar and arc source primitives using SOURC36 (which is assigned the next available element type number).

The macro is valid for use in 3D magnetic field analysis using a scalar potential formulation.

Current flows in a counterclockwise direction with respect to the working plane.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Excitation>Racetrack Coil

Main Menu>Preprocessor>Modeling>Create>Racetrack Coil

Main Menu>Solution>Define Loads>Apply>Magnetic>Excitation>Racetrack Coil

---

RADOPT, *--*, *FLUXTOL*, *SOLVER*, *MAXITER*, *TOLER*, *OVERRLEX*, *--*, *--*, *--*, *--*, *MAXFLUXITER*, *CONSERVATION*

Specifies Radiosity Solver options.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

--

Unused field.

FLUXTOL

Convergence tolerance for radiation flux. Defaults to 0.0001. This value is a relative tolerance.

SOLVER

Choice of solver for radiosity calculation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Gauss-Seidel iterative solver.

1

Direct solver.

2

Jacobi iterative solver (default).

MAXITER

Maximum number of iterations for the iterative solvers (*SOLVER* = 0 or 2). Defaults to 1000.

TOLER

Convergence tolerance for the iterative solvers (*SOLVER* = 0 or 2). Defaults to 0.1.

If *TOLER* $\geq$ 0, the value is interpreted as an absolute tolerance. If *TOLER* < 0, it is interpreted as a relative tolerance.

OVERRLEX

Over-relaxation factor applied to the iterative solvers (*SOLVER* = 0 or 2). Defaults to 0.1.

- -, - -, - -, - -

Unused fields

MAXFLUXITER

Maximum number of flux iterations to be performed according to the specified solver type:

0

If the FULL solver is specified (**THOPT (p. 1997)**,FULL), convergence criteria are monitored and iterations are performed until convergence occurs. If the QUASI solver is specified (**THOPT (p. 1997)**,QUASI), convergence criteria are ignored and one iteration is performed. This value is the default.

1, 2, 3, ...N

If the FULL solver is specified (**THOPT (p. 1997)**,FULL), convergence criteria are monitored and iterations are performed until convergence occurs, or until the specified number of iterations has been completed, whichever comes first. If the QUASI solver is specified (**THOPT (p. 1997)**,QUASI), convergence criteria are ignored and the specified number of iterations are completed.

To view *MAXFLUXITER* usage illustrations, see Figure : FULL Solution Method When Radiosity Is Present and Figure : QUASI Solution Method When Radiosity Is Present

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CONSERVATION

Key to account for the midside node temperature of underlying solid elements for radiosity calculations. Under normal circumstations using lower order elements, this option does not need to be activated. However, when using higher elements, you can improve energy conservation by setting *CONSERVATION* = 1.

0

Not active (default). The midside node temperatures are not accounted for in the radiosity calculations.

1

Active. The midside node temperatures are accounted for in the radiosity calculations.

Note:

To work effectively, *CONSERVATION* requires a one-to-one correspondance between the surface elements and their underlying solid elements. Therefore, it cannot be activated if the **RDEC (p. 1575)** command was issued when generating SURF251 or SURF252 elements.

Command Default

The Jacobi iterative solver (*SOLVER* = 2) is the default; it is only available for 3D models. If the analysis is 2D, the Gauss-Seidel iterative solver (*SOLVER* = 0) is used instead.

Notes

The radiation heat flux is linearized, resulting in robust convergence.

The radiation flux norm for *FLUXTOL* is expressed as:

$$\sqrt{\frac{\sum_{j=1}^{Nsurf}\left(Q_j^{i+1}-Q_j^{i}\right)^2}{\sum_{j=1}^{Nsurf}\left(Q_j^{i+1}\right)^2}}$$

where i is the pass or iteration number and j is the surface facet for radiation.

For a sufficiently small absolute tolerance value, relative tolerance converges in fewer iterations than absolute tolerance. For a sufficiently large absolute tolerance value, relative tolerance may cause convergence difficulties.

For more information about *FLUXTOL* and *MAXFLUXITER* usage, see Figure : FULL Solution Method When Radiosity Is Present and Figure : QUASI Solution Method When Radiosity Is Present in the *Thermal Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In Figure : FULL Solution Method When Radiosity Is Present and Figure : QUASI Solution Method When Radiosity Is Present (under Solving for Temperature and Radiosity in the *Thermal Analysis Guide*), refer to the $K_Q Q = F_Q$ equation system via the iterative method.

If *TOLER* $\geq$ 0, the iterative solver (*SOLVER* = 0 or 2) is converged for maximum value over a different *j* as shown:

$$\max_j \left| Q_j^{k+1} - Q_j^k \right| \leq TOLER$$

If *TOLER* < 0, the iterative solver (*SOLVER* = 0 or 2) is converged for maximum value over a different *j* as shown:

$$\max_j \frac{\left| Q_j^{k+1} - Q_j^k \right|}{\left| Q_j^{k+1} \right|} \leq \left| TOLER \right|$$

where:

j = number of radiation facets

k = number of iterations (*k* = 1 to *MAXITER*)

The Jacobi iterative solver (*SOLVER* = 2) is the only solver choice that runs in a fully distributed parallel fashion. Therefore, it is typically the best choice for optimal performance when running in distributed-memory parallel mode.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Solution Opt

Main Menu>Radiation Opt>Radiosity Meth>Solution Opt

Main Menu>Solution>Radiation Opts>Solution Opt

RAPPND, *LSTEP*, *TIME*

Appends results data from the database to the results file.

POST1 (p. 51): Load Case Calculations (p. 55)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LSTEP

Load step number to be assigned to the results data set. If it is the same as an existing load step number on the results file, the appended load step will be inaccessible. Defaults to 1.

TIME

Time value to be assigned to the results data set. Defaults to 0.0. A time value greater than the last load step should be used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is typically used to append the results from a load case combination to the results file. See the **LCWRITE (p. 1031)** command to create a separate load case file. Only summable and constant data are written to the results file by default; non-summable data are not written unless requested (**LCSUM (p. 1031)** command). RAPPND should not be used to append results from a harmonic analysis.

Menu Paths

Main Menu>General Postproc>Write Results

RATE, *Option*

Specifies whether the effect of creep strain rate will be used in the solution of a load step.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1571)

Option

Activates implicit creep analysis.

0 or OFF

No implicit creep analysis. This option is the default.

1 or ON

Perform implicit creep analysis.

Command Default

Issuing this command with no arguments deactivates implicit creep analysis.

Notes

Set *Option* = 1 (or ON) to perform an implicit creep analysis (**TB (p. 1899)**,CREEP with *TBOPT* ≥ 1). For viscoplasticity/creep analysis, *Option* specifies whether or not to include the creep calculation in the solution of a load step. If *Option* = 1 (or ON), the program performs the creep calculation. Set an appropriate time for solving the load step via a **TIME (p. 2000)**,*TIME* command.

Product Restrictions

This command works only when modeling *implicit* creep with either von Mises or Hill potentials.

When modeling implicit creep with von Mises potential, you can use the following elements: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLID272, SOLID273, SOLID285, SOLSH190, BEAM188, BEAM189, SHELL208, SHELL209, REINF264, SHELL281, and ELBOW290.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When modeling anisotropic creep (**TB (p. 1899)**,CREEP with **TB (p. 1899)**,HILL), you can also use the following elements: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, REINF264, REINF265, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, and ELBOW290.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Strn Rate Effect

Main Menu>Solution>Analysis Type>Sol'n Controls>Nonlinear

Main Menu>Solution>Load Step Opts>Nonlinear>Strn Rate Effect

/RATIO, *WN*, *RATOX*, *RATOY*

Distorts the object geometry.

GRAPHICS (p. 16): Scaling (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

RATOX

Distort object in the window X direction by this factor (defaults to 1.0).

RATOY

Distort object in the window Y direction by this factor (defaults to 1.0).

Command Default

No distortion.

Notes

Distorts the object geometry in a particular direction. An example of this command's use would be to allow long narrow sections to be distorted to a more square area for better display visualization.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RBE3, *INDEPEND*, *DOF*, *Depend*, *Wtfact*

Distributes the force/moment applied at an independent node to a set of dependent nodes.

PREP7 (p. 22): Constraint Equations (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

INDEPEND

Node at which the force/moment to be distributed will be applied (the independent node). To be included in the degree-of-freedom (DOF) solution, this node must be associated with an element.

DOF

Refers to the independent node DOFs to be used in the constraint equations. Valid labels are: UX, UY, UZ, ROTX, ROTY, ROTZ, UXYZ, RXYZ, ALL

Depend

The name of an array parameter that contains a list of dependent nodes. Must specify the starting index number. ALL can be used for the currently selected set of nodes. The dependent nodes must not be colinear (that is, must not be located on the same straight line). See "Notes" (p. 1573) for details.

Wtfact

The name of an array parameter that contains a list of weighting factors corresponding to each dependent node defined by *Depend*. Must have the starting index number. If not specified, the weighting factor for each dependent node defaults to 1.

Notes

RBE3 (p. 1573) distributes the force/moment applied at an independent node to a set of dependent nodes, taking into account the geometry of the dependent nodes as well as weighting factors. The force is distributed to the dependent nodes proportional to the weighting factors. The moment is distributed as forces to the dependent nodes; these forces are proportional to the distance from the center of gravity of the dependent nodes times the weighting factors. Only the translational degrees of freedom of the dependent nodes are used for constructing the constraint equations. Constraint equations are converted to distributed forces/moments on the dependent nodes during solution.

RBE3 (p. 1573) creates constraint equations such that the motion of the independent node is the average of the dependent nodes. For the rotations, a least-squares approach is used to define the "average rotation" at the independent node from the translations of the dependent nodes. If the dependent nodes are colinear, then one of the independent node rotations that is parallel to the colinear direction cannot be determined in terms of the translations of the dependent nodes. Therefore, the associated moment component on the independent node in that direction cannot be transmitted. When this case occurs, a warning message is issued and the constraint equations created by **RBE3 (p. 1573)** are ignored.

Applying this command to a large number of dependent nodes may result in constraint equations with a large number of coefficients. This may significantly increase the peak memory required during the process of element assembly. If real memory or virtual memory is not available, consider reducing the number of dependent nodes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the **DOFSEL (p. 569)** command to select the degrees of freedom (DOFs) of the dependent nodes to be included in the resulting constraint equations. (Be sure to issue **DOFSEL (p. 569)**,ALL *after* issuing **RBE3 (p. 1573)**.) This capability is useful if, for example, you want to ignore radial constraints in cylindrical geometries. The selected DOFs must collectively generate forces and moments on the independent node in all six DOFs.

RBE3 (p. 1573) is restricted to small-deflection analysis (large-deflection is not supported). As an alternative to **RBE3 (p. 1573)**, you can apply a similar type of constraint using contact elements and the internal multipoint constraint (MPC) algorithm. For more information, see Surface-based Constraints.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Coupling / Ceqn>Dist F/M at Indep

RCON

Specifies "Real constants" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Real Constants

RCYC, *IR*, *IA*, *SECTOR*, *Name*

Calculates cyclic results for a mode-superposition harmonic solution.

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previous variable, the previous variable will be overwritten with this result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IA

Reference number of the variable to be operated on.

SECTOR

Sector number to calculate the results for.

Name

Thirty-two character name identifying the variable on listings and displays. Embedded blanks are compressed for output.

Notes

This command calculates the harmonic response in the sector specified by *SECTOR* for the variable referenced by the reference number *IA*. Only component values for *IA* are valid (no principles or sums). The variable specified by *IR* will contain the harmonic solution. `Jobname.rfrq` from the cyclic mode-superposition harmonic solve and `Jobname.RST` or `Jobname.RSTP` from the cyclic modal solve must be available for the calculations to occur. The Jobname must be the same for the cyclic modal solve and the cyclic mode-superposition harmonic solve.

For *SECTOR* > 1, the result is in the nodal coordinate system of the base sector, and it is rotated to the expanded sector's location. Refer to Using the /CYCEXPAND Command in the *Cyclic Symmetry Analysis Guide* for more information.

See also Mode-Superposition Harmonic Cyclic Symmetry Analysis in the *Cyclic Symmetry Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

RDEC,*Option*,*REDUC*,*--*,*Nplace*

Defines the decimation parameters used by the radiosity solver method.

SOLUTION (p. 38): Radiosity (p. 50)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Command options:

DEFINE

Defines the decimation parameters (default).

STAT

Shows the status/listing. Other command options are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REDUC

Approximate reduction in the number of surface elements. Valid range is from 0.0 (no decimation, the default) to 1.0. This number is a factor applied to the initial number of element radiosity surfaces.

--

Unused field.

Nplace

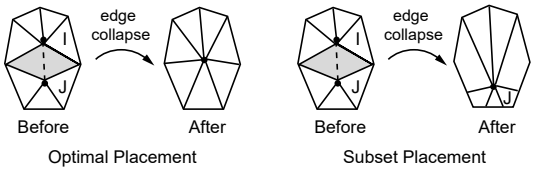
Node placement algorithm

OPTI

Optimal placement. An edge is collapsed by moving both nodes (I and J in the figure below) to a new location.

SUBS

Subset placement. An edge is collapsed by moving one node to another one. In the figure below, node I is moved to node J.



Notes

The **RDEC (p. 1575)** command sets decimation parameters. These parameters are used by the next **RSURF (p. 1631)** command to generate radiosity surface elements.

Decimation is the process of simplifying a fine surface mesh into a coarse one. This process is accomplished by a sequence of edge collapses.

The maximum degree of decimation (1.0) is unreachable. The real degree of decimation is always less than 1.0 because the decimated mesh must always consist of at least one element.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Decimation Options>Define Specifications

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Decimation Options>Define Specifications

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RDELE, *NSET1*, *NSET2*, *NINC*, *LCHK*

Deletes real constant sets.

PREP7 (p. 22): Real Constants (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET1*, *NSET2*, *NINC

Delete real constant sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL, ignore *NSET2* and *NINC* and all real constant sets are deleted.

LCHK

Specifies the level of element-associativity checking:

NOCHECK

No element-associativity check occurs. This option is the default.

WARN

When a section, material, or real constant is associated with an element, the program issues a message warning that the necessary entity has been deleted.

CHECK

The command terminates, and no section, material, or real constant is deleted if it is associated with an element.

Notes

Deletes real constant sets defined with the **R (p. 1565)** command.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>Add/Edit/Delete

Main Menu>Preprocessor>Real Constants

Main Menu>Preprocessor>Real Constants>Add/Edit/Delete

Main Menu>Solution>Load Step Opts>Other>Real Constants>Add/Edit/Delete

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

READ,*Fname*,*NSKIP*,*Format*,*Xfield*,*Yfield*,*Zfield*,*pRfield*,*pIfield*

Reads coordinate and pressure data from a file.

/MAP (p. 63): Pressure Mapping (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

NSKIP

Number of lines at the beginning of the file that will be skipped while it is read. Default = 0. *NSKIP* is ignored for *FileType* = CFXTBR or CFDPOST on the **FTYPE (p. 783)** command.

Format

For *FileType* = FORMATTED on the **FTYPE (p. 783)** command, *Format* is the read format in the FORTRAN FORMAT convention enclosed in parentheses; for example: (3e10.0,10x,e10.0,70x,e10.0)

Xfield*, *Yfield*, *Zfield*, *pRfield*, *pIfield

For *FileType* = CSV on the **FTYPE (p. 783)** command, these are field numbers locating the coordinates and real and imaginary (if present) pressures. The field value may not exceed 20.

Notes

Reads coordinate and pressure data from the specified file. The file type must have been previously specified on the **FTYPE (p. 783)** command.

Upon reading the file, nodes are created for the source points. For *FileType* = CFXTBR or CFDPOST on the **FTYPE (p. 783)** command, if face data is available, SURF154 elements are also created. A nodal component named SOURCENODES and an element component named SOURCEELEMS are created automatically.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REAL, *NSET*

Sets the element real constant set attribute pointer.

PREP7 (p. 22): Meshing (p. 29)
PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET

Assign this real constant set number to subsequently defined elements (defaults to 1).

Command Default

NSET = 1.

Notes

Identifies the real constant set number to be assigned to subsequently defined elements. This number refers to the real constant set number (*NSET*) defined with the real constant sets (**R (p. 1565)**). Real constant set numbers may be displayed (**/PNUM (p. 1457)**). If the element type requires no real constants, this entry is ignored. Elements of different type should not refer to the same real constant set.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

REALVAR, *IR*, *IA*, - -, - -, *Name*, - -, - -, *FACTA*

Forms a variable using only the real part of a complex variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

--,--

Unused fields.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields.

FACTA

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

Forms a variable using only the real part of a variable. Used only with harmonic analyses (**ANTYPE (p. 162)**,HARMIC).

Complex variables are stored in two-column arrays with the real component stored in the first column and the imaginary component stored in the second column. This command extracts the value stored in the first column (that is, real component). However with harmonic analyses, all variables are stored in two-column arrays as complex variables. If the variable is not complex, then the same value is stored in both columns. This command will extract the variable in the first column of the array, even if this variable is not the real component of a complex variable.

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Real Part

RECTNG,*X1*,*X2*,*Y1*,*Y2*

Creates a rectangular area anywhere on the working plane.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X1, X2

Working plane X coordinates of the rectangle.

Y1, Y2

Working plane Y coordinates of the rectangle.

Notes

The area will be defined with four keypoints and four lines. See the **BLC4 (p. 266)** and **BLC5 (p. 267)** commands for alternate ways to create rectangles.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Rectangle>By Dimensions

---

REMESH,*Action*,*Filename*,*Ext*,--,*Opt1*,*Opt2*

Specifies the starting and ending remeshing points, and other options, for rezoning.

SOLUTION (p. 38): Rezoning (p. 46)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

START

Starts the remeshing operation.

FINISH

Ends the remeshing operation.

READ

Reads in a generic (.cdb format) new mesh file generated by a third-party application. This remeshing option applies to both 2D and 3D rezoning.

SPLIT

Splits selected elements of an existing 2D or 3D mesh such that a quadrilateral element is split into four quadrilaterals, a degenerate quadrilateral is split into three quadrilaterals, and a quadratic triangular element is split into four quadratic triangles. A tetrahedral element is split into eight tetrahedra.

Filename

Name of a .cdb generic mesh file. The default value is *jobname*. Valid only when *Action* = READ.

Ext

File name extension. The only valid (and the default) extension is CDB. Valid only when *Action* = READ.

--

Unused field.

Opt1

Specifies options for the new mesh when using a generic imported mesh file or the mesh-splitting remeshing method. Valid only when *Action* = READ or *Action* = SPLIT.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REGE

Regenerates all node and element numbers on the new mesh using an offset of the highest existing node and element numbers. This is the default behavior when *Action* = READ; otherwise, this value is ignored.

KEEP

Keeps the similarly numbered nodes and elements in the new and the old meshes unchanged. Valid only when *Action* = READ.

TRAN

Generates transition elements to ensure nodal compatibility between split and unsplit parts of the mesh. Valid only when *Action* = SPLIT for 2D analyses.

Opt2

Specifies transition options for the mesh when elements are split. These options are valid only when *Action* = SPLIT for 2D analyses.

QUAD

Minimizes the number of degenerate elements in the transition mesh and tries to maximize the number of quadrilateral transition elements across several layers of elements from the split regions. This is the default behavior.

DEGE

Creates transition zones between the split and unsplit parts of the mesh using mostly degenerate elements with a single element layer.

Notes

This command is valid only during the rezoning (**REZONE (p. 1606)**) process.

In rezoning, **REMESH (p. 1581)**,START exits the solution processor temporarily and enters a special mode of the PREP7 preprocessor, after which a limited number of preprocessing commands are available for mesh control, but no solution commands are valid.

REMESH (p. 1581),FINISH exits the remeshing process and reenters the solution processor, at which point no preprocessing commands are available. If the new mesh exists, the command creates contact elements if needed, and transfers all boundary conditions (BCs) and loads from the original mesh to the new mesh. You can issue any list or plot command to verify the created contact elements, transferred BCs, and loads. **REMESH (p. 1581)**,FINISH is valid only after a previously issued **REMESH (p. 1581)**,START, and is the only way to safely end the remeshing operation (and exit the special mode of PREP7).

REMESH (p. 1581),READ is valid only when you want to perform a rezoning operation using a generic new mesh generated by a third-party application (rather than a new mesh generated internally by Mechanical APDL). The command is valid between **REMESH (p. 1581)**,START and **REMESH (p. 1581)**,FINISH. In this case, the only valid file extension is `.cdb` (*Ext* = CDB). When *Option* = KEEP, Mechanical APDL assumes that the common node and element numbers between the old and the new mesh are topolo-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

gically similar (that is, these commonly numbered areas have the same element connectivity and nodal coordinates).

REMESH (p. 1581),SPLIT is valid only when performing a rezoning operation via splitting the existing mesh. The command is valid between **REMESH (p. 1581)**,START and **REMESH (p. 1581)**,FINISH.

You can use **REMESH (p. 1581)**,READ and **REMESH (p. 1581)**,SPLIT for horizontal multiple rezoning provided that the meshes used in **REMESH (p. 1581)**,READ do not intersect. (Avoid issuing **AREMESH (p. 180)** after issuing either of these commands.)

For more information about the remeshing options available during rezoning, see Rezoning in the *Nonlinear Adaptivity Analysis Guide*.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

Main Menu>Solution>Manual Rezoning>Cancel

Main Menu>Solution>Manual Rezoning>Finish

Main Menu>Solution>Manual Rezoning>Read Mesh

Main Menu>Solution>Manual Rezoning>Select and Split Elems

Main Menu>Solution>Manual Rezoning>Start

***REMOVE**, *Name*, *Val1*, *Val2*, *Val3*

Suppresses rows or columns of a dense matrix or a vector.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name

Name of the matrix or vector to be revised.

Val1*, *Val2*, *Val3

Additional input. The meaning of *Val1* to *Val3* varies depending on the entity type (matrix or vector). See details below.

The following *Val1* through *Val3* fields are used if *Name* refers to a dense matrix created by the *DMAT (p. 551) command:

Val1

First row or column number to suppress.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val2

Last row or column number to suppress.

Val3

Specifies what to remove:

COL

Remove columns of the matrix (default).

ROW

Remove rows of the matrix.

The following *Val1* and *Val2* fields are used if *Name* refers to a vector created by the *VEC (p. 2085) command:

Val1

First value index to suppress.

Val2

Last value index to suppress.

Notes

The values of the original matrix or vector specified by *Name* are retained. The matrix or vector is resized to the new number of rows and columns.

Menu Paths

This command cannot be accessed from a menu.

***RENAME**,*OldName*,*NewName*,

Renames an existing vector or matrix.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

OldName

Name of the existing vector or matrix to be renamed.

NewName

New name for the vector or matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The ***RENAME (p. 1584)** command is used to rename APDL Math objects.

Menu Paths

This command cannot be accessed from a menu.

/RENAME,*Fname1*,*Ext1*,--,*Fname2*,*Ext2*,--,*DistKey*

Renames a file.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname1

The file to be renamed. You can also include an optional directory path as part of the specified file name; if not, the default file location is the working directory.

File name defaults to the current Jobname.

Ext1

Filename extension (eight-character maximum).

--

Unused field.

Fname2

The new name for the file. You can also include an optional directory path as part of the new file name; if not, the default is the working directory. A maximum of 248 characters is allowed for the file name (or combined file name and directory path, if both are specified).

Fname2 defaults to *Fname1*.

Ext2

Filename extension (eight-character maximum).

Ext2 defaults to *Ext1*.

--

Unused field.

DistKey

Key that specifies which rename operation is performed on all processes in distributed-memory parallel mode:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 (OFF or NO)

The program performs the rename operation only on the master process (default).

1 (ON or YES)

The program performs the rename operation locally on each process.

2 or BOTH

The program performs the rename operation for *Fname.Ext* on the master process and for *FnameN.Ext* on all processes.

Notes

Renames a file. Ex: **/RENAME (p. 1585)**,A,,,B renames file A to B in the same directory. **/RENAME (p. 1585)**,A,DAT,,,INP renames file A.DAT to A.INP. On all systems, this command will overwrite any existing file named B. See the *Operations Guide* for details. Only Mechanical APDL files should be renamed. Use **/SYS (p. 1895)** and system renaming commands for other files.

In distributed-memory parallel (DMP) mode, only the master process will rename *Fname1.Ext1* to *Fname2.Ext2* by default. However, when *DistKey* is set to 1 (or ON, or YES) or 2 (or BOTH), the command is executed by all processes. In this case, *Fname1* and *Fname2* will automatically have the process rank appended to them. This means *Fname1N.Ext1* will be renamed to *Fname2N.Ext2* by all processes, where *N* is the DMP process rank. For more information see Differences in General Behavior in the *Parallel Processing Guide*.

Renaming across system partitions may be internally done by a copy and delete operation on some systems.

This command is valid only at the Begin level.

Menu Paths

Utility Menu>File>File Operations>Rename

***REPEAT**, *NTOT*, *VINC1*, *VINC2*, *VINC3*, *VINC4*, *VINC5*, *VINC6*, *VINC7*, *VINC8*, *VINC9*, *VINC10*, *VINC11*

Repeats the previous command.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

NTOT

Number of times the preceding command is executed (including the initial execution). Must be 2 or greater. *NTOT* of 2 causes one repeat (for a total of 2 executions).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VINC1, VINC2, VINC3, ..., VINC11

Value increments applied to first through eleventh data fields of the preceding command.

Notes

***REPEAT (p. 1586)** must immediately follow the command that is to be repeated. The numeric arguments of the initial command may be incremented in the generated commands. The numeric increment values may be integer or real, positive or negative, zero or blank. Alphanumeric arguments cannot be incremented. For large values of *NTOT*, consider printout suppression (**/NOPR (p. 1251)** command) first.

Most commands beginning with slash (/), star (*), as well as "unknown command" macros, cannot be repeated. For these commands, or if more than one command is to be repeated, include them within a do-loop. File switching commands (those reading additional commands) cannot be repeated. If a ***REPEAT (p. 1586)** command immediately follows another ***REPEAT (p. 1586)** command, the repeat action only applies to the last non-***REPEAT (p. 1586)** command. Also, ***REPEAT (p. 1586)** should not be used in interactive mode immediately after a) a command (or its log file equivalent) that uses picking, or b) a command that requires a response from the user.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

/REPLOT, *Label*

Reissues the last display command.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Controls the type of replot.

RESIZE

Issued internally when a graphics window resize occurs (Default).

FAST

Only applicable for 3D devices that allow a fast redisplay for changes in the view characteristics only.

Notes

Reissues the last display command (**NPLOT (p. 1254)**, **EPLOT (p. 648)**, **KPLOT (p. 988)**, **PLNSOL (p. 1425)**, **PLVAR (p. 1444)**, etc.), along with its parameters, for convenience. The current display specifications are used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the last display command is invalid in a given processor, **/REPLOT (p. 1587)** is also invalid in that processor. However, if you attempt a **/REPLOT (p. 1587)** and the last display command is invalid in the current processor, Mechanical APDL generates an element display (**EPLOT (p. 648)**) instead, *as long as the last display command was* **PLNSOL (p. 1425)**, **PLESOL (p. 1401)**, *or* **PLDISP (p. 1400)**.

Example 12: /REPLOT (p. 1587) Replaced by EPLOT (p. 648) Automatically

PLNSOL (p. 1425), used to display solution results as continuous contours, is a valid command in the POST1 general postprocessor.

If you issue **PLNSOL (p. 1425)** followed by **/REPLOT (p. 1587)** while in POST1, **/REPLOT (p. 1587)** effectively reissues your earlier **PLNSOL (p. 1425)** command, along with its parameters.

If you then exit POST1, enter the PREP7 preprocessor, and again issue **/REPLOT (p. 1587)**, the program uses **EPLOT (p. 648)** internally instead.

The command substitution occurs because **PLNSOL (p. 1425)** is not a valid command in PREP7.

When you click one of the buttons on the Pan, Zoom, Rotate dialog box to manipulate the view of a model, **/REPLOT (p. 1587)** is issued internally. Thus, the substitution of **/REPLOT (p. 1587)** with **EPLOT (p. 648)** as described above may also occur for operations that you perform via with the Pan, Zoom, Rotate dialog box.

/REPLOT (p. 1587) does not show boundary conditions if they are applied only to a solid model and the last display command (such as **EPLOT (p. 648)**) displays the finite element model. To show boundary conditions, the following options are available:

- Issue **/REPLOT (p. 1587)** after you issue **SBCTRAN (p. 1647)** to transfer solid model boundary conditions to the finite element model.
- Issue **/REPLOT (p. 1587)** after you issue a solid model display command (such as **VPLOT (p. 2147)**).

This command is valid in any processor (except as noted above).

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>General Postproc>Path Operations>Delete Path>All Paths

Main Menu>General Postproc>Path Operations>Delete Path>By Name

Main Menu>General Postproc>Path Operations>Plot Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Scale Icon

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Delete Path>All Paths

Main Menu>Preprocessor>Path Operations>Delete Path>By Name

Main Menu>Preprocessor>Path Operations>Plot Paths

Utility Menu>Plot>Replot

Utility Menu>PlotCtrls>Style>Symmetry Expansion>Expansion by values

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RESCOMBINE,*NUMFILES*,*Fname*,*Ext*,*Lstep*,*Sbstep*,*Fact*,*KIMG*,*TIME*,*ANGLE*,*NSET*,*ORDER*

Reads results from local results files into the database after a distributed-memory parallel solution.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NUMFILES

Number of local results files that are to be read into the database from the distributed-memory parallel solution. This number should be equal to the number of processes used in the parallel solution.

Fname

File name (jobname) used during the distributed parallel solution. The file name must be an alphanumeric string (up to 32 characters) enclosed in single quotes.

Ext

File extension for the results files (for example, RST, RTH, RMG, etc.). The file extension must be an alphanumeric string (up to 8 characters) enclosed in single quotes.

Lstep

Load step number of the data set to be read (defaults to 1):

N

Read load step *N*.

FIRST

Read the first data set (*Sbstep* and *TIME* are ignored).

LAST

Read the last data set (*Sbstep* and *TIME* are ignored).

NEXT

Read the next data set (*Sbstep* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

PREVIOUS

Read the previous data set (*Sbstep* and *TIME* are ignored). If at the first data set, the last data set will be read as the previous.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NEAR

Read the data set nearest to *TIME* (*Sbstep* is ignored). If *TIME* is blank, read the first data set.

LIST

Scan the results files and list a summary of each load step (*KIMG*, *TIME*, *ANGLE*, *NSET*, and *ORDER* are ignored.)

Sbstep

Substep number within *Lstep* (defaults to the last substep of the load step). For a buckling (**ANTYPE (p. 162)**,BUCKLE) or modal (**ANTYPE (p. 162)**,MODAL) analysis, *Sbstep* corresponds to the mode number (defaults to the first mode). Specify *Sbstep* = LAST to store the last substep for the specified load step.

If *Lstep* = LIST, *Sbstep* = 0 or 1 lists the basic step information; *Sbstep* = 2 also lists the basic step information, but includes the load step title, and labels imaginary data sets if they exist.

Fact

Scale factor applied to data read from the files. If zero (or blank), a value of 1.0 is used. A nonzero factor excludes non-summable items. Harmonic velocities or accelerations may be calculated from the displacement results from a modal (**ANTYPE (p. 162)**,MODAL) or harmonic (**ANTYPE (p. 162)**,HARMIC) analysis. If *Fact* = VELO, the harmonic velocities (v) are calculated from the displacements (d) at a particular frequency (f) according to the relationship $v = 2\pi fd$. Similarly, if *Fact* = ACEL, the harmonic accelerations (a) are calculated as $a = (2\pi f)^2 d$.

KIMG

Used only with complex results (harmonic and complex modal analyses).

0 or REAL

Store the real part of a complex solution (default).

1, 2 or IMAG

Store the imaginary part of a complex solution.

TIME

Time-point identifying the data set to be read. For a harmonic analysis, time corresponds to the frequency. For a buckling analysis, time corresponds to the load factor. Used only in the following cases: If *Lstep* = NEAR, read the data set nearest to *TIME*. If both *Lstep* and *Sbstep* are zero (or blank), read data set at time = *TIME*. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results file (**OUTRES (p. 1336)**) for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, the last time point will be used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANGLE

Circumferential location (0.0 to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (PLANE25, PLANE75, PLANE78, PLANE83, and SHELL61) of the load case. See the *Mechanical APDL Theory Reference* for details. Note that factored values of applied constraints and loads will overwrite any values existing in the database.

NSET

Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *Sbstep*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **RESCOMBINE (p. 1589)**,,,,LIST.

ORDER

Key to sort the harmonic index results. This option applies to cyclic symmetry buckling and modal analyses only, and is valid only when *Lstep* = FIRST, LAST, NEXT, PREVIOUS, NEAR or LIST.

ORDER

Sort the harmonic index results in ascending order of eigenfrequencies or buckling load multipliers.

(blank)

No sorting takes place.

Notes

RESCOMBINE (p. 1589) is a Mechanical APDL command macro enabling you to combine results from a distributed-memory parallel (DMP) solution. A character string input for any argument must be enclosed in single quotes (for example, 'FIRST' input for *Lstep*).

In a distributed memory parallel solution, a global results file is saved by default. However, if you issued **DMPOPTION (p. 559)**,RST,NO in the parallel solution, no global results file is written and all local results files will be kept. In this case, you can use **RESCOMBINE (p. 1589)** in the general postprocessor (**/POST1 (p. 1461)**) to read results into the database for postprocessing.

RESCOMBINE (p. 1589) cannot be used to combine results from local files generated during a distributed-memory parallel solution that used the frequency or cyclic harmonic index domain decomposition method (**DDOPTION (p. 508)**,FREQ or **DDOPTION (p. 508)**,CYCHI).

To use **RESCOMBINE (p. 1589)**, all local results files from the distributed-memory parallel solution must be in the current working directory. If running on a single machine, the local results files are saved in the working directory by default. If running on a cluster, the local results files are kept in the working directory on each compute node. For the latter case, copy the local results files to the working directory on the primary compute node.

Similar to **SET (p. 1724)**, **RESCOMBINE (p. 1589)** defines the data set to be read from the results files into the database. Various operations can also be performed during the read operation. (See **SET (p. 1724)** for more information.) The database must have the model data available (or issue **RESUME (p. 1601)** before **RESCOMBINE (p. 1589)** to restore the geometry from `Jobname.db`).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After issuing **RESCOMBINE (p. 1589)** to combine a set of data into the database, you can issue **RESWRITE (p. 1604)** to write the data set into a new results file. The new results file will essentially contain the current set of results data for the entire (that is, global) model.

Upon completion of a **RESCOMBINE (p. 1589)** operation, the current file for postprocessing (**FILE (p. 752)**) is set to the last local results file specified via **RESCOMBINE (p. 1589)**. For example, if reading in four local results files, the results file for POST1 is specified as `Jobname3.RST` when **RESCOMBINE (p. 1589)** is complete. Therefore, be aware that some downstream postprocessing actions (such as **SET (p. 1724)**) may be operating on only this one local results file.

RESCOMBINE (p. 1589) is intended for use in POST1. If you want to postprocess DMP solution results using the POST26 time-history postprocessor, combine your local results files into one global results file (**DMPOPTION (p. 559)**,RST,YES).

The load case commands in the general postprocessor (such as **LCDEF (p. 1021)**, **LCFILE (p. 1023)**, **LCOPER (p. 1026)**, etc.) are not supported when using **RESCOMBINE (p. 1589)**. Those commands set up pointers in the results file used for postprocessing; they cannot be used with the local results files used by **RESCOMBINE (p. 1589)**.

/CYCEXPAND (p. 454), which performs a cyclic expansion, cannot be used with **RESCOMBINE (p. 1589)**.

Menu Paths

This command cannot be accessed from a menu.

RESCONTROL,*Action*,*Ldstep*,*Frequency*,*MAXFILES*,*--*,*MAXTotalFiles*,*Filetype*

Controls file writing for multiframe restarts.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Command action:

DEFINE

Specifies how often *.xnnn* restart files (default) or *.rdnn* remeshing database files (for nonlinear mesh adaptivity analysis) are written for a load step. The file type controlled by this command is determined by *Filetype*.

FILE_SUMMARY

Prints the substep and load-step information for all *.xnnn* or *.rdnn* files for the current jobname in the current directory. If specified, all other arguments are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STATUS

Issuing the command lists the current status in the tables of restart controls specified previously by **RESCONTROL (p. 1592)**. If this option is specified, all other arguments are ignored.

NORESTART

Cleans up some of the restart files after a distributed-memory parallel (DMP) solution. (Not valid for nonlinear mesh adaptivity.)

The master process will not have the following files in the working directory at the end of the run: `.esav`, `.osav`, `.xnnn`, `.rdb`, `.ldhi`. The worker processes will not have the following files in the working directory at the end of the run: `.esav`, `.osav`, `.xnnn`, `.rst` (or `.rth`, etc.). Some restart files are never written, some are removed upon exiting the solution processor (for example, upon **FINISH (p. 758)**), and some are removed upon exiting the program.

This option is useful for cleaning up files written by all distributed processes, especially when you know that these restart files will not be needed later. If this option is specified, all other arguments are ignored.

If this option is used in a shared-memory parallel (SMP) environment, most restart files in the working directory are removed. It has the same effect as issuing **RESCONTROL (p. 1592)**,,NONE.

LINEAR

Same as *Action* = DEFINE, but for linear static applications only. For a linear static analysis, the restart capability is normally not needed; however, it is typically needed when a subsequent linear perturbation analysis is desired. By default, none of the restart files are written for a linear static analysis.

DELETE

Delete the restart control specification corresponding to the *Ldstep* label on a previous **RESCONTROL (p. 1592)**,DEFINE command.

Ldstep

Specifies how the `.xnnn` or `.rdnn` files are written for the specified load steps. This option also affects how often the load history information is written to the `.ldhi` file.

ALL

Write the `.xnnn` or `.rdnn` files at the same substep *Frequency* for all load steps; write the load history information to the `.ldhi` file for all load steps. For `.rdnn` files, this option is the default.

LAST

Write the `.xnnn` or `.rdnn` files for the last load step only; write load-history information to the `.ldhi` file for the last load step only. This option is the default for nonlinear static and full transient analyses for `.xnnn` files. If specified, all remaining arguments are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N

Number that indicates how often the .*xnnn* or .rd*nn* files are written.

Input a positive number to write the .*xnnn* or .rd*nn* files at the substep *Frequency* indicated only for load step *N*. Other load steps will be written at the default substep frequency or at a frequency defined by a previous **RESCONTROL (p. 1592)** specification. Load-history information is written to the .ldhi file only for load steps *N*.

Specifying a negative number (-*N*) is valid for controlling .*xnnn* files only. The files are written for every *N*th load step at the specified substep *Frequency*. The load history information is written to the .ldhi file every *N*th load step. This option is suitable for restart applications in which more than a few hundred load steps are required. Compared to the ALL and positive *N* options, it can save disk space, as the .ldhi file is smaller and fewer .*xnnn* files are written.

If *Ldstep* = -*N*, all other *Ldstep* options specified by **RESCONTROL (p. 1592)** are ignored and the program follows the -*N* option (write load history information every *N*th load step). If you want to change this pattern, issue **RESCONTROL (p. 1592)**,DELETE, -*N*, then issue another **RESCONTROL (p. 1592)** command with the desired *Ldstep* option.

NONE

No multiframe restart files (.rdb, .ldhi, .*xnnn*) are created. (Not valid for nonlinear mesh adaptivity analysis.)

This option is the default for mode-superposition analyses. The remaining arguments are ignored.

For nonlinear static, linear static, and full transient analyses, this option enables a restart to occur at the last or abort point (using the .emat, .esav or .osav, and .db files).

For mode-superposition transient analyses, this option allows a restart from the last point using the .rdsp and .db files.

Frequency

Frequency at which the .*xnnn* files are written at the substep level, or at which the .rd*nn* files are written at the remeshing level:

NONE

Do not write .*xnnn* files for this load step (not available for .rd*nn* files).

LAST

Write the .*xnnn* files for the last substep of the load step only (default for nonlinear static and full transient analyses), or write the .rd*nn* files for the last remesh of the load step only (default for nonlinear mesh adaptivity analysis).

N

If *N* is positive, write the .*xnnn* files at every *N*th substep of a load step, or write the .rd*nn* files at every *N*th remesh of a load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *N* is negative (not valid for .rd*nn* files), write *N* equally spaced .*xnnn* files within a load step.

In nonlinear static and full transient analyses, -*N* is valid only when automatic time stepping is enabled (**AUTOTS (p. 207)**,ON).

In mode-superposition analyses, negative *N* is always valid.

MAXFILES

Maximum number of .*xnnn* files to save within a load step (not available for .rd*nn* files):

-1

Overwrite existing .*xnnn* files (default). The total maximum number of .*xnnn* files for one run is 999. If this number is reached before the analysis is complete, the program will reset the .*xnnn* file numbering back to 1 and continue to write .*xnnn* files; the program keeps the newest 999 restart files and overwrites the oldest restart files. For this option, the maximum number of files can be changed to a number less than 999 by setting the *MAXTotalFiles* argument.

0

Do not overwrite any existing .*xnnn* files. The total maximum number of .*xnnn* files for one run is 999. If this number is reached before the analysis is complete, the analysis continues but no longer writes any .*xnnn* files.

N

Maximum number of .*xnnn* files to keep for each load step. When *N* .*xnnn* files have been written for a load step, the program overwrites the first .*xnnn* file of that load step for subsequent substeps. *N* must be <= 999. If a total of 999 restart files is reached before the analysis is complete, the analysis continues but writes no additional .*xnnn* files.

--

Reserved for future use.

MAXTotalFiles

Total number of restart files to keep. Default = 999 for .*xnnn* files and 99 for .rd*nn* files. This option is valid only when *MAXFILES* = -1 (default).

Valid *MAXTotalFiles* values are 1 through 999 for .*xnnn* files, and 1 through 99 for .rd*nn* files.

When the total number of restart files written exceeds *MAXTotalFiles*, the program resets the .*xnnn* or .rd*nn* file numbering back to 1 and continues to write .*xnnn* or .rd*nn* files. The newest files are retained and the oldest files are overwritten.

The *MAXtotalFiles* value specified applies to all subsequent load steps. To reset it to the default, reissue the command with *MAXTotalFiles* = 0 or some negative value.

If *MAXTotalFiles* is set to different values at different load steps, and if the value of *MAXTotalFiles* specified in the prior load step is larger than that of the current load step, the program can

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

only overwrite the current number of maximum restart files up to the number *MAXTotalFiles* currently specified (which is smaller than the previous number).

The recommended way to control the maximum number of restart files is to specify *MAXTotalFiles* at the first load step and not vary it in subsequent load steps. Also, *MAXTotalFiles* is best used when *Ldstep* = -*N* or ALL.

Filetype

The type of restart file to be controlled by this command. Valid only when *Action* = DEFINE:

XNNN

Control .*xnnn* files (default).

RDNN

Control .rd*nn* remeshing database files. Needed only for a nonlinear mesh adaptivity analysis.

Command Default

If the **RESCONTROL (p. 1592)** command is not issued during a structural analysis, the .rdb and .ldhi files are written as described in Restarting an Analysis.

For nonlinear static and full transient analysis:

The default behavior is multiframe restart: **RESCONTROL (p. 1592)**,DEFINE,LAST,LAST

The .*xnnn* file is written at the last substep of the last load step. An .r*nnn* file is also written at the iteration prior to the abort point of the run if a jobname.abt file was used (or the **Stop** button was pressed in the GUI), or if the job terminated because of a failure to reach convergence or some other solution error. No information at the aborted substep is saved to the .*xnnn* file.

For nonlinear mesh adaptivity analysis:

The default behavior for .rd*nn* files written is: **RESCONTROL (p. 1592)**,DEFINE,ALL,LAST,,,,RDNN

The .rd*nn* file is written at the last remesh of every load step by default.

The .rd*nn* and .r*nnn* files interact with each other. Generally, .rd*nn* file writing is superior to that of .r*nnn* file writing. For example, if no **RESCONTROL (p. 1592)**,DEFINE command is issued, the default behavior is that both .rd*nn* and .r*nnn* files are written at the last occurrence of every load step (equivalent to **RESCONTROL (p. 1592)**,DEFINE,ALL,LAST and **RESCONTROL (p. 1592)**,DEFINE,ALL,LAST,,,,RDNN).

Notes

RESCONTROL (p. 1592) sets up the restart parameters for a multiframe restart, enabling you to restart an analysis from any load step and substep for which there is an .*xnnn* file. You can perform a multiframe restart for static and transient (full or mode-superposition method) analyses only. For more information about multiframe restarts and descriptions of the contents of the files used, see Restarting an Analysis in the *Basic Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Syntax

Multiframe restart files are generically indicated here as .*xnnn* files. They correspond to .r*nnn* files for nonlinear static and full transient analyses, and .m*nnn* files for mode-superposition analyses.

Remeshing database files are indicated as .rd*nn* files. This type of restart file is needed only after remeshing during a nonlinear mesh adaptivity analysis.

When *Action* = DEFINE, the specified *Filetype* determines the type of file (.*xnnn* or .rd*nn*) controlled by this command.

Number of Restart Files Allowed

The total number of restart files for any analysis cannot exceed 999 (for example, jobname.r001 to jobname.r999).

The total number of remeshing database files cannot exceed 99 (for example, jobname.rd01 to jobname.rd99).

Considerations for Nonlinear Mesh Adaptivity Analysis

To control *both* .*xnnn* and .rd*nn* file writing (*Filetype* = XNNN and *Filetype* = RDNN, respectively), separate **RESCONTROL (p. 1592)** commands are necessary.

Action = NORESTART and *Ldstep* = NONE are not valid and will cause the analysis to fail.

Ldstep = *-N* is not valid for controlling .*xnnn* files.

Limiting the Number of Files Saved

If you have many substeps for each load step and are writing .*xnnn* files frequently, you may want to set *MAXFILES* to limit the number of .*xnnn* files saved, as they can fill your disk quickly.

You can specify *MAXFILES* and *Frequency* for individual load steps. These arguments take on the default value or the value defined by **RESCONTROL (p. 1592)**,,ALL,*Frequency*,*MAXFILES* if they are not explicitly defined for a specific load step.

When .*xnnn* files are written over many load steps, you may want to further limit the number of .*xnnn* files by setting *MAXTotalFiles*.

Maximum Number of Load Steps

You can specify a maximum of ten load steps; that is, you can issue the **RESCONTROL (p. 1592)**,,N command a maximum of ten times. Specified load steps cannot be changed in a restart.

Specifying *Ldstep* = LAST or *-N*

The program accepts only one occurrence of **RESCONTROL (p. 1592)** with *Ldstep* = LAST. If you issue **RESCONTROL (p. 1592)**,,LAST,*Frequency*,*MAXFILES* multiple times, the last specification overwrites the previous one.

The program accepts only one occurrence of **RESCONTROL (p. 1592)** with a negative *Ldstep* value (**RESCONTROL (p. 1592)**,,*N* where *N* is a negative number). If you issue **RESCONTROL (p. 1592)**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

multiple times with a negative *Ldstep* value, the last specification overwrites the previous one.

Using RESCONTROL (p. 1592) in a Restarted Analysis

The **RESCONTROL (p. 1592)** command is not valid in the restarted load step of a restart analysis. It is only valid in subsequent load steps.

Example Usage

Multiframe Restart

Linear Perturbation Analysis

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Sol'n Options

Main Menu>Preprocessor>Loads>Load Step Opts>Nonlinear>Restart Control

Main Menu>Solution>Analysis Type>Sol'n Controls>Sol'n Options

Main Menu>Solution>Load Step Opts>Nonlinear>Restart Control

RESET

Resets all POST1 or POST26 specifications to initial defaults.

POST1 (p. 51): Set Up (p. 51)
POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Has the same effect as entering the processor the first time within the run. In POST1, resets all specifications to initial defaults, erases all element table items, path table data, and load case pointers. In POST26, resets all specifications to initial defaults, erases all variables defined, and zeroes the data storage space.

Menu Paths

Main Menu>General Postproc>Reset

Main Menu>TimeHist Postpro>Reset Postproc

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/RESET

Resets display specifications to their initial defaults.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Resets slash display specifications (**/WINDOW (p. 2177)**, **/TYPE (p. 2034)**, **/VIEW (p. 2131)**, etc.) back to their initial default settings (for convenience). Also resets the focus location to the geometric center of the object.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Reset Plot Ctrls

RESP,*IR*,*LFTAB*,*LDTAB*,*specType*,*dampRatio*,*DTIME*,*TMIN*,*TMAX*,*inputType*

Generates a response spectrum.

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the response spectrum results (2 to NV (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with these results.

LFTAB

Reference number of variable containing frequency table (created with **FILLDATA (p. 756)** or **DATA (p. 497)** command). The frequency table defines the number and frequency of oscillating systems used to determine the response spectrum. The frequency interval need not be constant over the entire range. Frequencies must be input in ascending order.

LDTAB

Reference number of variable containing the input time-history.

specType

Defines the type of response spectrum to be calculated:

0 or 1

Displacement (relative to base excitation)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2

Velocity (relative to base excitation)

3

Acceleration response spectrum (absolute)

4

Pseudo-velocity

5

Pseudo-acceleration

dampRatio

Ratio of viscous damping to critical damping (input as a decimal number).

DTIME

Integration time step. This value should be equal to or greater than the integration time step used in the initial transient analysis performed to generate the input time-history (*LDTAB*).

DTIME defaults to a value of 1/(20*FMAX), where FMAX is the highest frequency in the frequency table (*LFTAB*) except when the time-history is read from the reduced displacement (RDSP) file following a mode-superposition transient analysis without an expansion pass. In this case, *DTIME* defaults to the time step value used in the analysis.

TMIN,TMAX

Specifies a subset of the input time-history (*LDTAB*) to be used in the response spectrum calculation. Defaults to the full time range.

inputType

Defines the type of the input time-history:

0

Displacement (default)

1

Acceleration

Notes

This command generates a response spectrum from a displacement or acceleration time-history and frequency data. The response spectrum is defined as the maximum response of single degree of freedom systems of varying frequency (or period) to a given input support excitation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A response spectrum analysis (**ANTYPE (p. 162)**, SPECTR with **SPOPT (p. 1837)**, SPRS or MPRS) requires a response spectrum input. This input can be determined from the response spectrum printout or display of this command.

If a response spectrum is to be calculated from a given displacement (or acceleration) time-history, the displacement time-history may be input to a single one-element reduced linear transient dynamic (**ANTYPE (p. 162)**,TRANS) analysis, so that the calculated output (which should be the same as the input) will be properly located on the file.

The integration time step (argument *DTIME* on the **RESP (p. 1599)** command) and the damping coefficient (argument *dampRatio*) are constant over the frequency range. The number of calculations done per response spectrum curve is the product of the number of input solution points (*TMAX*-*TMIN*)/*DTIME* and the number of frequency points (frequencies located in variable *LFTAB*).

Input solution points requested (using *DTIME* and the frequency range) at a time not corresponding to an actual displacement solution time on the file are linearly interpolated with respect to the existing points.

For the theory of the response spectrum calculation, see POST26 - Response Spectrum Generator (RESP).

For an example of the command usage, see Generating a Response Spectrum in the *Basic Analysis Guide*.

Menu Paths

Main Menu>TimeHist Postpro>Generate Spectrm

RESUME,*Fname*,*Ext*,--,*NOPAR*,*KNOPLOT*

Resumes the database from the database file.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to DB if *Fname* is blank.

--

Unused field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NOPAR

Parameter resume key:

0

All data in the database, *including* the scalar parameters, are replaced with the data saved on `Jobname.db` (default).

1

All data in the database, *except* the scalar parameters, are replaced with the data saved on `Jobname.db`.

KNOPLOT

If equal to 1, will suppress automatic plot. Otherwise, if the GUI is on and this **RESUME (p. 1601)** command was not read from a file, the selected elements from *Fname* are plotted. (If there are no selected elements, selected nodes are plotted. If no nodes, volumes; if no volumes, areas; if no areas, lines; if no lines, keypoints. If there are no selected keypoints, the screen is erased.)

Notes

The **RESUME (p. 1601)** command resumes a database file into Mechanical APDL. The command causes the database file (`Jobname.db`) to be read, thereby resetting the database (including any geometry settings) either a) as it was at the last **SAVE (p. 1645)** command, or b) as it was saved with the last **/EXIT (p. 708)** command, whichever was last.

For multiple load step analyses (because only the data for one load step at a time may reside in the database), the load step data restored to the database will correspond to the load step data written when the save occurred.

If the database file was saved in another Ansys, Inc. product, it may contain element type and **KEYOPT (p. 979)** specifications which are invalid in the resuming product. Immediately after the database resume is completed, you should redefine these invalid element types and **KEYOPT (p. 979)** settings to valid ones (**ET (p. 686)**, **KEYOPT (p. 979)**).

The *NOPAR* = 1 option should *not* be used if array parameters are defined, as existing array parameters might be redefined with arbitrary values. For a more general method of preventing the replacement of both scalar and array parameters, see **PARSAV (p. 1352)** and **PARRES (p. 1351)**.)

If a radiosity mapping data file (`.rsm` file) was saved by the previous **SAVE (p. 1645)** command, that mapping file must be present in the directory along with the database file in order for radiosity surface elements (SURF251, SURF252) to be correctly mapped onto the model when **RESUME (p. 1601)** is issued.

This command is valid in any processor. If used in the solution processor, this command is valid only within the first load step.

Menu Paths

Utility Menu>File>Resume from

Utility Menu>File>Resume Jobname.db

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RESVEC, *KeyVect*, --, --, --, *KeyResp*

Calculates or includes residual vectors or residual responses

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KeyVect

Residual vector key:

OFF

Do not calculate or include residual vectors (default).

ON

Calculate or include residual vectors.

--, --, --

Unused fields.

KeyResp

Residual response key:

OFF

Do not calculate or include residual responses (default).

ON

Calculate or include residual responses.

Command Default

No residual quantities are calculated or included in the analysis.

Notes

In a modal analysis, the **RESVEC (p. 1603)** command calculates residual vectors (or responses). In a mode-superposition transient, mode-superposition harmonic, PSD or spectrum analysis, the command includes residual vectors. The command must be issued during the first modal solve.

In the expansion pass of a mode-superposition transient or mode-superposition harmonic analysis, the command includes residual responses.

In a component mode synthesis (CMS) generation pass, the **RESVEC (p. 1603)** command calculates one residual vector which is included in the normal modes basis used in the transformation matrix. It is supported for the three available CMS methods. **RESVEC (p. 1603)**,ON can only be specified in the first load step of a generation pass and is ignored if issued at another load step.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If rigid-body modes exist, pseudo-constraints are required for the calculation. Issue the **D (p. 483)**,,,SUPPORT command to specify only the minimum number of pseudo-constraints necessary to prevent rigid-body motion.

Both residual vector and residual response approaches cannot be used in the same analysis.

For more information about residual vector or residual response formulation, see Residual Vector Method or Residual Response Method in the *Mechanical APDL Theory Reference*.

Menu Paths

This command cannot be accessed from a menu.

RESWRITE, *Fname*, --, --, --, *cFlag*

Appends results data from the database to a results file.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name extension varies as follows:

- `.rst` for structural, fluid, or coupled-field analyses
- `.rth` for thermal or electrical analyses
- `.rmg` for magnetic analyses

--, --, --

Unused fields.

cFlag

0

The complex results flag is set to 0 in the results file header. This is the default option.

1

The complex results flag is set to 1 in the results file header.

Notes

The **RESWRITE (p. 1604)** command appends a data set to the specified file by writing the results data currently in the database. If the intended results file does not exist, it will be created and will include

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the geometry records. The current load step, substep, and time (or frequency) value are maintained. All data (summable and nonsummable) are written.

When complex results are appended, *cFlag* must be set to 1 so that the header is consistent with the results written on the file.

The command is primarily intended for use in a top-down substructuring analysis, where the full model is resumed and the results data read from the use pass results file (**SET (p. 1724)**), and subsequently from all substructure expansion pass results files (**APPEND (p. 173)**). The full set of data in memory can then be written out via the **RESWRITE (p. 1604)** command to create a complete results file (as though you had run a nonsubstructured analysis).

The **RESWRITE (p. 1604)** command can also be used to write a global results file for a distributed-memory parallel (DMP) solution. This should only be necessary if the **RESCOMBINE (p. 1589)** command was used to combine results from local results files into the database. The **RESWRITE (p. 1604)** command can then be used to write the combined results into a new results file. This new results file will essentially contain the current set of results data for the entire (that is, global) model.

Menu Paths

This command cannot be accessed from a menu.

---

*RETURN, *Level*

Returns input stream to a higher level.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Level

Number of levels to move up from the current level.

Negative --

Move relative to current level. For example: *Return,-2 will go up two levels from the current level.

Positive --

Move to absolute level. For example: *Return,2 will go to level 2.
Level 0 is the primary input file.

Notes

This command is used to jump to the macro call sequence, ending the current macro file, and returning to the line after the calling line in the previous file. Unlike the ***GO (p. 864)** command, this command may be used inside ***IF (p. 927)** or ***DO (p. 567)** constructs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

REZONE, *Option*, *LDSTEP*, *SBSTEP*

Initiates the rezoning process, sets rezoning options, and rebuilds the database.

SOLUTION (p. 38): Rezoning (p. 46)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

The rezoning method to employ:

MANUAL

Manual rezoning. You decide when to use rezoning, what region(s) to rezone, and what remeshing method to use on the selected region(s). This method is currently the default and only option.

LDSTEP

The load step number at which rezoning should occur. The default value is the highest load step number found in the Jobname.R*nnn* files (for the current jobname and in the current directory).

SBSTEP

The substep number of the specified load step (*LDSTEP*) at which rezoning should occur. The default value is the highest substep number found in the specified load step in the Jobname.R*nnn* files (for the current jobname and in the current directory).

Notes

The **REZONE (p. 1606)** command rebuilds the database (.db file) based on the specified load step and substep information, and updates nodes to their deformed position for remeshing.

Before issuing this command, clear the database via the **/CLEAR (p. 350)** command.

For more information, see Rezoning in the *Advanced Analysis Guide*.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

Main Menu>Solution>Manual Rezoning>Start

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RFORCE, *NVAR*, *NODE*, *Item*, *Comp*, *Name*

Specifies the total reaction force data to be stored.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number assigned to this variable (2 to NV (**NUMVAR (p. 1302)**)). Overwrites any existing results for this variable.

NODE

Node for which data are to be stored. If *NODE* = P, graphical picking is enabled (valid only in the GUI).

Item

Label identifying the item. Valid item labels are shown in the table below. Some items also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

Name

Thirty-two character name identifying the item on printouts and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

Defines the total reaction force data (static, damping, and inertial components) to be stored from single pass (**ANTYPE (p. 162)**,STATIC or TRANS) solutions or from the expansion pass of mode-superposition (**ANTYPE (p. 162)**,HARMIC or TRANS) solutions.

Table 225: RFORCE - Valid Item and Component Labels

Valid item and component labels for node results are:		
Item	**Comp**	**Description**
F	X,Y,Z	X, Y, or Z structural force
M	X,Y,Z	X, Y, or Z structural moment
HEAT[1]		Heat flow
FLOW		Fluid flow
AMPS		Current flow
FLUX		Magnetic flux
CSG	X,Y,Z	X, Y, or Z magnetic current segment component
RATE		Diffusion flow rate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid item and component labels for node results are:

Item	Comp	Description
VLTG		Voltage drop
CURT		Current
CHRG		Charge

1. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . ., HTOP instead of HEAT.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

/**RGB**,*Keyword*,*PRED*,*PGRN*,*PBLU*,*N1*,*N2*,*NINC*,*NCNTR*

Specifies the RGB color values for indices and contours.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Keyword

Determines how RGB modifications will be applied.

INDEX

Specifies that subsequent color values apply to Mechanical APDL color indices (0-15).

CNTR

Specifies that subsequent color values apply to contours (1-128). Applies to C-option devices only (i.e. X11C or Win32C).

PRED

Intensity of the color red, expressed as a percentage.

PGRN

Intensity of the color green, expressed as a percentage.

PBLU

Intensity of the color blue, expressed as a percentage.

N1

First index (0-15), or contour (1-128) to which the designated RGB values apply.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N2

Final index (0-15), or contour (1-128) to which the designated RGB values apply.

NINC

The step increment between the values *N1* and *N2* determining which contours or indices will be controlled by the specified RGB values.

NCNTR

The new maximum number of contours (1-128).

Notes

Issuing the **/CMAP (p. 359)** command (with no filename) will restore the default color settings.

Menu Paths

This command cannot be accessed from a menu.

RIGID,*Dof1*,*Dof2*,*Dof3*,*Dof4*,*Dof5*,*Dof6*

Specifies known rigid body modes (if any) of the model.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Dof1*,*Dof2*,*Dof3*,...,*Dof6

Up to six global Cartesian directions of the rigid modes. For a completely free 2D model, use ALL or UX, UY, ROTZ. For a completely free 3D model, use ALL or UX, UY, UZ, ROTX, ROTY, ROTZ. For a constrained model, use UX, UY, UZ, ROTX, ROTY, or ROTZ, as appropriate, to specify each and every unconstrained direction which exists in the model (not specifying every direction may cause difficulties in extracting the modes).

Notes

Specifies known rigid body modes (if any) of the model. This command applies only to a component mode synthesis (CMS) analysis (see the **CMSOPT (p. 378)** command). Any rigid body modes specified must be permitted by the applied displacement constraints (that is, do not specify a rigid body mode in a constrained direction). Reissue the command to redefine the specification. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

RIGRESP, *Option*, *Method*, *Val1*, *Val2*

Specifies the rigid response calculation.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Flag to activate or deactivate the rigid response calculation:

1 (ON or YES)

Activate.

2 (OFF or NO)

Deactivate. This value is the default.

Method

Method used to calculate the rigid response:

GUPTA

Gupta method.

LINDLEY

Lindley-Yow method.

Val1

If *Method* = GUPTA, *Val1* represents the frequency F_1 in Hertz.

If *Method* = LINDLEY, *Val1* is the Zero Period Acceleration (ZPA). If a scale factor is defined (FACT in the **SVTYP (p. 1887)** command), it is used to scale this value

Val2

If *Method* = GUPTA, *Val2* represents the frequency F_2 in Hertz.

Notes

This rigid response calculation is only valid for single-point response spectrum analysis (**SPOPT (p. 1837)**, SPRS) and multiple-point response spectrum analysis (**SPOPT (p. 1837)**, MPRS) with combination methods

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

square root of sum of squares (**SRSS (p. 1845)**), complete quadratic (**CQC (p. 433)**) or Rosenblueth (**ROSE (p. 1615)**).

This command is also valid in PREP7.

Additional Information

Performing a Single-Point Response Spectrum (SPRS) Analysis in the *Structural Analysis Guide*

Performing a Multi-Point Response Spectrum (MPRS) Analysis in the *Structural Analysis Guide*

Rigid Responses in the *Mechanical APDL Theory Reference*

MMASS (p. 1131) command

Menu Paths

This command cannot be accessed from a menu.

RLIST,*NSET1*,*NSET2*,*NINC*

Lists the real constant sets.

PREP7 (p. 22): Real Constants (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET1*, *NSET2*, *NINC

List real constant sets from *NSET1* to *NSET2* (defaults to *NSET1*) in steps of *NINC* (defaults to 1). If *NSET1* = ALL (default), ignore *NSET2* and *NINC* and list all real constant sets (**R (p. 1565)**).

Notes

The real constant sets listed contain only those values specifically set by the user. Default values for real constants set automatically within the various elements are not listed.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Properties>All Real Constants

Utility Menu>List>Properties>Specified Real Constants

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/RMDIR, *Dir*

Removes (deletes) a directory.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Dir

The directory to remove. If no path is provided, it will be assumed to be in the current working directory. All files in the directory are also removed.

Notes

Removes a directory on the computer on which Mechanical APDL is currently running. No warning or prompt is given, so use with extreme caution.

Menu Paths

This command cannot be accessed from a menu.

RMFLVEC

Writes eigenvectors of fluid nodes to a file for use in damping parameter extraction.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

RMFLVEC (p. 1612) extracts the modal information from the modal results file for all nodes specified in a node component called 'FLUN'. This component should include all nodes which are located at the fluid-structural interface. Mode shapes, element normal orientation, and a scaling factor are computed and stored in a file `Jobname`.EFL. For damping parameter extraction, use the **DMPEXT (p. 557)** command macro. See Introduction for more information on thin film analyses.

FLUID136 and FLUID138 are used to model the fluid interface. Both the structural and fluid element types must be active. The fluid interface nodes must be grouped into a component 'FLUN'. A results file of the last modal analysis must be available.

Menu Paths

Main Menu>General Postproc>ThinFilm>Extract Eigv

Main Menu>Solution>ThinFilm>DampExtract>Eigenfrequency

Main Menu>Solution>ThinFilm>DampExtract>Frequency Range

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>ThinFilm>RayleighDamp

RMODIF, *NSET*, *STLOC*, *V1*, *V2*, *V3*, *V4*, *V5*, *V6*

Modifies real constant sets.

PREP7 (p. 22): Real Constants (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSET

Number of existing real constant set to be modified.

The labels CONT and GCN are also valid input for defining or modifying real constants associated with contact elements (see Notes (p. 1613)).

STLOC

Starting location in table for modifying data. For example, if *STLOC* = 1, data input in the *V1* field is the first constant in the set. If *STLOC* = 7, data input in the *V1* field is the seventh constant in the set, etc. Must be greater than zero.

V1

New value assigned to constant in location *STLOC*. If zero (or blank), a zero value will be assigned.

V2, V3, V4, . . . , V6

New values assigned to constants in the next five locations. If blank, the value remains unchanged.

Notes

Allows modifying (or adding) real constants to an existing set (**R (p. 1565)**) at any location.

Specify *NSET* = CONT to define or modify real constants for all real constant sets associated with contact elements in pair-based contact definitions. Specify *NSET* = GCN to define or modify real constants for real constant sets that were previously assigned by the **GCDEF (p. 788)** command; that is, real constants used in general contact interactions.

This command is also valid in SOLUTION. For important information about using this command within the solution phase, see What Are Nonstandard Uses? in the *Advanced Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RMORE, *R7*, *R8*, *R9*, *R10*, *R11*, *R12*

Adds real constants to a set.

PREP7 (p. 22): Real Constants (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

R7, R8, R9, . . . , R12

Add real constants 7 to 12 (numerical values or table names) to the most recently defined set.

Notes

Adds six more real constants to the most recently defined set. Repeat the **RMORE (p. 1614)** command for constants 13 to 18, again for 19-24, etc.

If using table inputs (SURF151, SURF152, FLUID116, CONTA172, CONTA174, CONTA175, and CONTA177 only), enclose the table name in % signs (for example, *%tabname%*).

When copying real constants to new sets, Ansys, Inc. recommends that you use the command input. If you do use the GUI, restrict the real constant copy to only the first six real constants (real constants seven and greater will be incorrect for both the master and copy set).

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Rotary

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Mechanical>Spring>Nonlin Trans

Main Menu>Preprocessor>Modeling>Create>Circuit>Builder>Transducer>ElecMech

Main Menu>Preprocessor>Modeling>Create>Circuit>Edit Real Cnst

ROCK, *CGX*, *CGY*, *CGZ*, *OMX*, *OMY*, *OMZ*

Specifies a rocking response spectrum.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

CGX, CGY, CGZ

Global Cartesian X, Y, and Z location of center of rotation about which rocking occurs.

OMX, OMY, OMZ

Global Cartesian angular components of the rocking.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Specifies a rocking response spectrum effect in the spectrum (**ANTYPE (p. 162)**,SPECTR) analysis.

The excitation direction with rocking included is not normalized to one; rather, it scales the spectrum. For example, consider a node at coordinates (1,1,0), subject to an excitation in the X direction (*SEDX* = 1.0 on **SED (p. 1704)**), and a rocking with center *CGX* = 1.0, *CGY* = *CGZ* = 0, and angular component about Z (*OMZ* = 0.5). The total excitation direction at this node is:

$$\{D\}=\begin{Bmatrix}1\\0\\0\end{Bmatrix}+\begin{Bmatrix}0\\0\\0.5\end{Bmatrix}\times\left(\begin{Bmatrix}1\\1\\0\end{Bmatrix}-\begin{Bmatrix}1\\0\\0\end{Bmatrix}\right)=\begin{Bmatrix}0.5\\0\\0\end{Bmatrix}$$

So that half the spectrum input is applied at this node.

For more information on the equations, see Participation Factors and Mode Coefficients.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings

ROSE, *SIGNIF*, *Label*, *TD*, *ForceType*

Specifies the Rosenblueth mode combination method.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIGNIF

Combine only those modes whose significance level exceeds the SIGNIF threshold. For single-point, multi-point, or DDAM response spectrum (*Spectype* = SPRS, MPRS, or DDAM on **SPOPT (p. 1837)**), the significance level of a mode is defined as the mode coefficient divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and does not contribute to the mode combinations. The higher the SIGNIF threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If SIGNIF is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT (p. 1837)**,PSD.)

Label

Label identifying the combined mode solution output.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc. are available.

TD

Time duration for earthquake or shock spectrum. *TD* defaults to 10.

ForceType

Label identifying the forces to be combined:

STATIC

Combine the modal static forces (default).

TOTAL

Combine the modal static plus inertial forces.

Notes

For more information on spectrum analysis combination methods, see Combination of Modes.

Modes whose effective damping ratio defined in Equation 14.42 exceeds or equals 1.0 are not combined.

This command is also valid in PREP7.

Menu Paths

This command cannot be accessed from a menu.

RPOLY,*NSIDES*,*LSIDE*,*MAJRAD*,*MINRAD*

Creates a regular polygonal area centered about the working plane origin.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSIDES

Number of sides in the regular polygon. Must be greater than 2.

LSIDE

Length of each side of the regular polygon.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MAJRAD

Radius of the major (or circumscribed) circle of the polygon. Not used if *LSIDE* is input.

MINRAD

Radius of the minor (or inscribed) circle of the polygon. Not used if *LSIDE* or *MAJRAD* is input.

Notes

Defines a regular polygonal area on the working plane. The polygon will be centered about the working plane origin, with the first keypoint defined at $\theta = 0°$. The area will be defined with *NSIDES* keypoints and *NSIDES* lines. See the **RPR4 (p. 1617)** and **POLY (p. 1460)** commands for other ways to create polygons.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Circumscr Rad

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Inscribed Rad

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>By Side Length

---

RPR4, *NSIDES*, *XCENTER*, *YCENTER*, *RADIUS*, *THETA*, *DEPTH*

Creates a regular polygonal area or prism volume anywhere on the working plane.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NSIDES

The number of sides in the polygon or prism face. Must be greater than 2.

XCENTER, YCENTER

Working plane X and Y coordinates of the center of the polygon or prism face.

RADIUS

Distance (major radius) from the center to a vertex of the polygon or prism face (where the first keypoint is defined).

THETA

Angle (in degrees) from the working plane X-axis to the vertex of the polygon or prism face where the first keypoint is defined. Used to orient the polygon or prism face. Defaults to zero.

DEPTH

The perpendicular distance (either positive or negative based on the working plane Z direction) from the working plane representing the depth of the prism. If *DEPTH* = 0 (default), a polygonal area is created on the working plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines a regular polygonal area anywhere on the working plane or prism volume with one face anywhere on the working plane. The top and bottom faces of the prism are polygonal areas. See the **RPOLY (p. 1616)**, **POLY (p. 1460)**, **RPRISM (p. 1618)**, and **PRISM (p. 1497)** commands for other ways to create polygons and prisms.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Hexagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Octagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Pentagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Septagon
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Square
Main Menu>Preprocessor>Modeling>Create>Areas>Polygon>Triangle
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Hexagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Octagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Pentagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Septagonal
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Square
Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>Triangular

---

RPRISM, *Z1*, *Z2*, *NSIDES*, *LSIDE*, *MAJRAD*, *MINRAD*

Creates a regular prism volume centered about the working plane origin.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Z1, Z2

Working plane Z coordinates of the prism.

NSIDES

Number of sides in the polygon defining the top and bottom faces of the prism. Must be greater than 2.

LSIDE

Length of each side of the polygon defining the top and bottom faces of the prism.

MAJRAD

Radius of the major (or circumscribed) circle of the polygon defining the top and bottom faces of the prism. Not used if *LSIDE* is input.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MINRAD

Radius of the minor (or inscribed circle) of the polygon defining the top and bottom faces of the prism. Not used if *LSIDE* or *MAJRAD* is input.

Notes

Defines a regular prism volume centered about the working plane origin. The prism must have a spatial volume greater than zero. (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) The top and bottom faces are polygonal areas that are parallel to the working plane but neither face need be coplanar with (that is, "on") the working plane. The first keypoint defined for each face is at $\theta = 0°$. See the **RPR4 (p. 1617)** and **PRISM (p. 1497)** commands for other ways to create prisms.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Circumscr Rad

Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Inscribed Rad

Main Menu>Preprocessor>Modeling>Create>Volumes>Prism>By Side Length

---

RPSD, *IR*, *IA*, *IB*, *ITYPE*, *DATUM*, *Name*, --, *SIGNIF*

Calculates response power spectral density (PSD).

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previous variable, the previous variable will be overwritten with this result.

IA, IB

Reference numbers of the two variables to be operated on. If only one, leave *IB* blank.

ITYPE

Defines the type of response PSD to be calculated:

0,1

Displacement (default).

2

Velocity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3

Acceleration.

DATUM

Defines the reference with respect to which response PSD is to be calculated:

1

Absolute value.

2

Relative to base (default).

Name

Thirty-two character name identifying variable on listings and displays. Embedded blanks are compressed for output.

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. The significance level is defined as the modal covariance matrix term divided by the maximum of all the modal covariance matrix terms. Any term whose significance level is less than *SIGNIF* is considered insignificant and does not contribute to the response. All modes are taken into account by default (*SIGNIF* = 0.0).

The significance level definition is identical to the one used for the combination (*SIGNIF* on the **PSDCOM (p. 1527)** command); however, the default value is different.

The significance does not apply to spatial correlation (**PSDSPL (p. 1532)**) and wave propagation (**PSDWAV (p. 1534)**) response power spectral density.

Notes

This command calculates response power spectral density (PSD) for the variables referenced by the reference numbers *IA* and *IB*. The variable referred by *IR* will contain the response PSD. You must issue the **STORE (p. 1867)**,PSD command first; `File.psd` must be available for the calculations to occur.

See POST26 - Response Power Spectral Density in the *Mechanical APDL Theory Reference* for more information on these equations.

Menu Paths

Main Menu>TimeHist Postpro>Calc Resp PSD

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RSMESH,*P0*,*RID*,*KCN*,*Kdir*,*VALUE*,*NDPLANE*,*PSTOL*,*Ecomp*

Generates a result section.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P0

User-defined anchor point node number. This must be a previously defined target element pilot node (see "Notes" (p. 1622) for more information). If *P0* is specified, *RID*, *KCN*, *Kdir*, *VALUE*, and *NDPLANE* are ignored. The result section is defined in the x-direction of the nodal coordinate system of the pilot node.

If *P0* is not specified, the program computes the geometry center point as the anchor point location by default.

RID

Unique real constant ID number that has not been previously assigned to any other elements. If *RID* is not specified, the program selects the next available real constant ID number.

KCN

Coordinate system number used to define the result surface and normal direction.

Kdir

Direction (x, y, or z) normal to the result surface in the *KCN* coordinate system.

VALUE

Point along the *Kdir* axis at which to locate the result surface. Ignored if *NDPLANE* is specified.

NDPLANE

Node number of existing node used to locate the result surface. If *NDPLANE* is specified, the location of the result surface is defined by the *Kdir* coordinate of *NDPLANE* in the *KCN* coordinate system.

PSTOL

Optional tolerance below *VALUE*. Allows nodes occurring precisely at or slightly below the result surface to be identified properly as above the plane. Has the effect of shifting the plane down by *PSTOL*. The following expression represents the default value:

$$\frac{\sqrt{\Delta X^2+\Delta Y^2+\Delta Z^2}}{1000}$$

where ΔX, ΔY, and ΔZ are the dimensions of the locally selected region of the model based on nodal locations (that is, ΔX = X*max* - X*min*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ecomp

If specified, the name of a component to be composed of elements underneath the result section generated by the **RSMESH (p. 1621)** command.

Notes

This command defines a result section and automatically imbeds contact elements (CONTA172 or CONTA174) on the surface of the selected base elements. See Monitoring Result Section Data During Solution in the *Structural Analysis Guide* for more information on how to use this command.

A user-specified anchor point and local coordinate system can be specified by defining a pilot node (a target element defined using **TSHAP (p. 2023)**,PILO) before issuing the **RSMESH (p. 1621)** command. Input the pilot node number for *P0*. Only one pilot node should be associated with a result section; the pilot node should not be used for any other purpose (such as remote loading). The contact elements generated for the result section will have the same real constant ID as the pilot node target element.

Menu Paths

This command cannot be accessed from a menu.

RSOPT,*Opt*,*Filename*,*Ext*,*Dir*

Creates or loads the radiosity mapping data file for SURF251 or SURF252 element types.

SOLUTION (p. 38): Radiosity (p. 50)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Opt

File option:

SAVE

Write the radiosity mapping data to a file. (Default)

LOAD

Read in the specified mapping data file.

Fname

File name for radiosity mapping data file. Defaults to Jobname.

Ext

Filename extension for radiosity mapping data file (default = .rsm).

Dir

Directory path for radiosity mapping data file. If you do not specify a directory path, it will default to your working directory.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Use this command to manually create or load a radiosity mapping data file. This command is useful if you want to create the mapping data file without issuing **SAVE (p. 1645)** or **CDWRITE (p. 293)**, or if you want to specify that the file be located in a directory other than your working directory. Also use this command to manually load an existing mapping data file during a restart.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

This command cannot be accessed from a menu.

RSPLIT,*Option*,*Label*,*Name1*,*Name2*,*Name3*,*Name4*,*Name5*,*Name6*,*Name7*,*Name8*,*Name9*,*Name10*,*Name11*,*Name12*,*Name13*,*Name14*,*Name15*,*Name16*

Creates one or more results file(s) from the current results file based on subsets of elements.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Specify which results to include for the subset of elements.

ALL

Write all nodal and element results based on the subset of elements (default).

EXT

Write only the nodal and element results that are on the exterior surface of the element subset. The results data will be averaged as in PowerGraphics (see **AVRES (p. 212)**) when this results file is brought into POST1. Only valid for solid elements.

Label

Define where the element subset is coming from.

ALL

Use all selected element components (**CMSEL (p. 374)**) (default).

ESEL

Use the currently selected (**ESEL (p. 661)**) set of elements. *Name1* defines the results file name.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LIST

Use *Name1* to *Name16* to list the element component and/or assembly names (that contain element components).

Name1*, *Name2*, *Name3*, . . . , *Name16

Up to 16 element component and/or assembly names (that contain element components).

Command Default

Write all data available for the element subset.

Notes

Results files will be named based on the element component or assembly name, for example, *Cname*.rst, except for the ESEL option, for which you must specify the results file name (no extension) using the *Name1* field. Note that the .rst filename will be written in all uppercase letters (CNAME.rst) (unless using the ESEL option); when you read the file, you must specify the filename using all uppercase letters (that is, file,CNAME). You may repeat the **RSPLIT (p. 1623)** command as often as needed. All results sets on the results file are processed. Use **/AUX3 (p. 209)** to produce a results file with just a subset of the results sets.

Use **INRES (p. 953)** to limit the results data written to the results files.

The subset geometry is also written so that no database file is required to postprocess the subset results files. You must not resume any database when postprocessing one of these results files. The input results file must have geometry written to it (that is, do not use **/CONFIG (p. 410)**,NORSTGM,1).

Applied forces and reaction forces are not apportioned if their nodes are shared by multiple element subsets. Their full values are written to each results file.

Each results file renumbers its nodes and elements starting with 1.

This feature is useful when working with large models. For more information on the advantages and uses of the **RSPLIT (p. 1623)** command, see Splitting Large Results Files in the *Basic Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

RSTCONTROL, *Type*, *Cname*, *Method*, *MethodItem*

Controls whether element single value results are written to the results file.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Solution format for database and file-write control:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AUTO

Write element output quantities (STRS, EPEL, EPPL, EPCR, EPTH) in single value form for those element types that support it, and in element nodal form for all other element types (default). See Element Single-Value Results in the *Element Reference* for a list of supported element types.

ELSV

Write element output quantities only in single value form. If the single value form is not supported by the element types in the model, none of the applicable results (STRS, EPEL, EPPL, EPCR, EPTH) will be available.

ELND

Write all element output quantities only in element nodal form.

ERASE

Reset **RSTCONTROL (p. 1624)** specifications to their default values.

STAT

List the current **RSTCONTROL (p. 1624)** specifications.

Cname

Name of the component (created via **CM (p. 356)**) defining the selected set of elements for which this specification is active. If no name is entered, the specification applies to all elements.

Method

For single value results, *Method* controls how the integration point values of each element are reduced to one value:

AVG

Use a simple average of values from all integration points (default).

MAXE

Use only the integration point corresponding to the maximum equivalent (EQV) stress or strain quantity specified by *MethodItem*.

MAXP

Use only the integration point corresponding to the maximum 1st principal stress or strain quantity specified by *MethodItem*.

MAXS

Use only the integration point corresponding to the maximum shear stress or strain quantity specified by *MethodItem*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MethodItem

The quantity used to determine the integration point where single value results are reported. Only valid when *Method* = MAXE, MAXP, or MAXS.

STRS

Choose the integration point based on stress.

EPEL

Choose the integration point based on elastic strain.

EPPL

Choose the integration point based on plastic strain.

EPCR

Choose the integration point based on creep strain.

EPTH

Choose the integration point based on thermal strain.

Command Default

The program default is to write element nodal results for all elements.

Notes

RSTCONTROL (p. 1624) is an optional output control command that specifies whether element nodal results or element single value results are written to the results file. This command works in conjunction with the **OUTRES (p. 1336)** command. **OUTRES (p. 1336)** specifies when each element quantity will be output, while **RSTCONTROL (p. 1624)** specifies the type of result (element nodal versus single value). Not all elements support single value results. For more information, see Element Single-Value Results in the *Element Reference*.

The element output quantities affected by this command are:

stress (STRS)

elastic strain (EPEL)

plastic strain (EPPL)

creep strain (EPCR)

thermal/swelling strains (EPTH)

You can issue up to 50 **RSTCONTROL (p. 1624)** commands in an analysis. **RSTCONTROL (p. 1624)**,ERASE erases the existing output specifications and resets the counted number of **RSTCONTROL (p. 1624)** commands to zero.

RSTCONTROL (p. 1624) is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

RSTMAC,*File1*,*Lstep1*,*Sbstep1*,*File2*,*Lstep2*,*Sbstep2*,--,*MacLim*,*Cname*,*KeyPrint*

Calculates modal assurance criterion (MAC/FRF) and matches nodal solutions from two results files or from one results file and one universal format file.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

File1

File name (248 characters maximum) corresponding to the first results file (`.rst` or `.rstp` file). If the file name does not contain the extension, it defaults to `.rst`.

Lstep1

Load step number of the results to be read in *File1*.

N

Reads load step *N*. Defaults to 1.

Sbstep1

Substep number of the results to be read in *File1*.

N

Reads substep *N*. Only valid for MAC.

All

Reads all substeps. This value is the default.

File2

File name (248 characters maximum) corresponding to the second file (`.rst`, `.rstp`, or `.unv` file). If the file name does not contain the extension, it defaults to `.rst`.

Lstep2

Load step number of the results to be read in *File2*.

N

Reads load step *N*. Defaults to 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sbstep2

Substep number of the results to be read in *File2*.

N

Reads substep *N*. Only valid for MAC.

All

Reads all substeps. This value is the default.

--

Unused field.

MacLim

Smallest acceptable criterion value. Must be ≥ 0 and ≤ 1. The default value is 0.90.

Cname

Name of the component from the first file (*File1*). The component must be based on nodes. If unspecified, all nodes are matched and used for MAC calculations. If a component name is specified, only nodes included in the specified component are used. Not applicable to node mapping (*Option* = NODMAP on **MACOPT (p. 1098)**).

KeyPrint

Printout options:

0

Printout matched solutions table. This value is the default.

1

Printout matched solutions table and full MAC table.

2

Printout matched solutions table, full MAC table and matched nodes table.

Notes

The **RSTMAC (p. 1627)** command allows the comparison of the solutions from either:

- Two different results files. Valid only for MAC calculations.
- One result file (`.rst`) and one universal format file (`.unv`).

Either the modal assurance criterion (MAC) or the frequency response function (FRF) correlation method is used. (For more details see POST1 - Modal Assurance Criterion (MAC) and POST1 – Frequency response function correlation in the *Theory Reference*.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The meshes read from *File1* and *File2* may be different. The nodes of *File1* are matched with the nodes of *File2* based on either node location (default) or node number. The solutions are compared for the identified pair of matched nodes. The nodes can also be mapped and the solutions interpolated from *File1*. See the **MACOPT (p. 1098)** command for all options.

Units and coordinate systems must be the same for both models.

Results may be real or complex; however, if results in *File1* have a different type than results in *File2*, only the real parts of the solutions are taken into account in MAC calculations. The analysis type can be arbitrary for MAC while only harmonic analysis is supported for FRF criteria calculations.

Non-structural degrees of freedom can be considered. Degrees of freedom can vary between *File1* and *File2*, but at least one common degree of freedom must exist.

The solutions read on the results files are not all written to the database; therefore, subsequent plotting or printing of solutions is not possible. A **SET (p. 1724)** command must be issued after the **RSTMAC (p. 1627)** command to post-process each solution.

The corresponding database file (`.db`) for `File1` on **RSTMAC (p. 1627)** must be resumed before running the command in the following cases:

- A component (*Cname*) is used on **RSTMAC (p. 1627)**.
- The nodes are mapped (**MACOPT (p. 1098)**,NODMAP,YES).
- The nodes are matched using a relative tolerance (**MACOPT (p. 1098)**,RELTOLN).

RSTMAC (p. 1627) comparison on cyclic symmetry analysis works only if the number of sectors on *File1* and *File2* are the same, and the database is saved ***after*** the solution is finished. Also, a comparison cannot be made between cyclic symmetry results and the full 360 degree model results (*File1* – cyclic solution, *File2* – full 360 degree model solution). Comparing cyclic symmetry solutions written on a selected set of nodes (**OUTRES (p. 1336)**) is not supported.

The modal assurance criterion values can be retrieved as parameters using the ***GET (p. 797)** command (*Entity* = **RSTMAC (p. 1627)**).

FRF correlation is only supported for the comparison of an `.rst` and a `.unv` file. All substeps are considered for both files to define the frequency domain for criteria calculations. The printout options consist of listing the table of the criterion results for each frequency in the frequency domain (*KeyPrint* = 0, 1, or 2) and the matched nodes table (*KeyPrint* = 2).

Example Usage

For a detailed discussion on using RSTMAC with examples see Comparing Nodal Solutions From Two Models (**RSTMAC (p. 1627)**) in the *Basic Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RSTOFF, *Lab*, *OFFSET*

Offsets node or element IDs in the FE geometry record.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

The offset type:

NODE

Offset the node IDs.

ELEM

Offset the element IDs.

OFFSET

A positive integer value specifying the offset value to apply. The value must be greater than the number of nodes or elements in the existing superelement results file.

Command Default

Issuing the **RSTOFF (p. 1630)** command with no specified argument values applies no offsets.

Notes

The **RSTOFF (p. 1630)** command offsets node or element IDs in the FE geometry record saved in the `.rst` results file. Use the command when expanding superelements in a bottom-up substructuring analysis (where each superelement is generated individually in a generation pass, and all superelements are assembled together in the use pass).

With appropriate offsets, you can write results files with unique node or element IDs and thus display the entire model even if the original superelements have overlapping element or node ID sets. (Such results files are incompatible with the `.db` database file saved at the generation pass.)

The offset that you specify is based on the original superelement node or element numbering, rather than on any offset specified via a **SESYMM (p. 1718)** or **SETRAN (p. 1730)** command. When issuing an **RSTOFF (p. 1630)** command, avoid specifying an offset that creates conflicting node or element numbers for a superelement generated via a **SESYMM (p. 1718)** or **SETRAN (p. 1730)** command.

If you issue the command to set non-zero offsets for node or element IDs, you must bring the geometry into the database via the **SET (p. 1724)** command so that Mechanical APDL can display the results. Specify appropriate offsets to avoid overlapping node or element IDs with other superelement results files.

The command is valid only in the first load step of a superelement expansion pass.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

RSURF,*Options*,*Delopts*,*ETNUM*

Generates the radiosity surface elements and stores them in the database.

SOLUTION (p. 38): Radiosity (p. 50)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Options

Command options:

CLEAR

Deletes radiosity surface elements and nodes. The set of elements and nodes to be deleted is defined by *Delopts*. *ETNUM* is ignored.

DEFINE

Creates the radiosity surface elements and nodes (default).

STAT

Shows the status/listing. Other command options are ignored.

Delopts

Deletion options

ALL

Deletes all radiosity surface elements and nodes.

LAST

Deletes radiosity surface elements and nodes created by the last **RSURF (p. 1631)** command.

ETNUM

Element type number. Leave blank to indicate the next available number.

Notes

This command generates the radiosity surface elements (SURF251, SURF252) based on the **RSYMM (p. 1632)** and **RDEC (p. 1575)** parameters and stores them in the database. It works only on the faces of selected underlying elements that have RDSF flags on them and all corner nodes selected. You can issue multiple **RSURF (p. 1631)** commands to build the radiosity model. However, all **RSURF (p. 1631)** commands must be issued after issuing the **RSYMM (p. 1632)** command, and after the model is complete (that is, after all meshing operations are complete).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you do issue multiple **RSURF (p. 1631)** commands for different regions, you must first mesh the different regions, and then generate the radiosity surface elements on each meshed region individually. Use **RSURF (p. 1631)**,,,ETNUM to assign a separate element type number to each region. This procedure allow you to identify the individual regions later in the multi-field analysis.

If the underlying solid elements are higher order, the radiosity surface elements are always lower order (4- or 3-node in 3D or 2-node in 2D). Decimation will always occur before any symmetry operations.

For 2D axisymmetric YR models, the newly-generated nodes can have only positive Y coordinates.

The **RSURF (p. 1631)** command assigns real constant set number 1 to all SURF251 and SURF252 elements generated, irrespective of the current real constant set attribute pointer (**REAL (p. 1579)** command). If the generated elements require a real constant set other than number 1, you must manually change the set number for those elements by using the **EMODIF (p. 626)**,,REAL command.

If you have already issued **RSURF (p. 1631)** for a surface and you issue **RSURF (p. 1631)** again, the program creates a new set of radiosity surface elements and nodes over the existing set, resulting in an erroneous solution.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

This is an action command (that creates or deletes surface meshes) and is serial in nature. Even if a DMP solution is running, the **RSURF (p. 1631)** command runs serially.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Clear Radiation Surface Mesh

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Generate Radiation Surface Mesh

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Clear Radiation Surface Mesh

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Surface Mesh>Generate Radiation Surface Mesh

---

RSYMM,*Option*,*CS*,*Axis*,*NSECT*,*CONDVALUE*,*SVAL*,*EVAL*

Defines symmetry, rotation, or extrusion parameters for the radiosity method.

SOLUTION (p. 38): Radiosity (p. 50)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Command options:

CLEAR

Deletes all symmetry/extrusion definitions. Other command options are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEFINE

Defines the symmetry/extrusion definition (default).

STAT

Shows the status/listing. Other command options are ignored.

COND

Activates or deactivates condensation for all defined radiation symmetries/extrusions, which reduces the size of the radiosity equation system (see Figure 14: Usage Example:*Option* = COND (p. 1638)). Default is off.

Note:

Condensation via **RSYMM (p. 1632)**,COND is not recommended as the most efficient solution for symmetric models. To best leverage model symmetry to improve efficiency, use view factor condensation via the **VFCO (p. 2093)** command, which condenses the view factor matrix in addition to simplifying the radiosity equations (see Considerations for View Factor Condensation (p. 1636) for details).

CS

Local coordinate system (≥11) as defined using the **LOCAL (p. 1059)** or **CS (p. 435)** commands or the global coordinate system (0). For planar reflection, the coordinate system origin must be on the plane of symmetry (POS) and one of its axes must be normal to the POS. For cyclic reflection, the coordinate system origin must be coincident with the center of rotation (COR). Only Cartesian systems are valid.

Axis

Axis label of the coordinate system (*CS*) that is normal to the POS for planar reflection, or label to indicate the type of extrusion. For cyclic reflection, this field must be blank, and it is assumed that the Z axis is aligned with the axis of rotation.

X, Y, or Z

Planar reflection. For 2D model planar reflections, valid labels are X or Y. For 3D model planar reflections, valid labels are X, Y, or Z.

ZEXT

Linear extrusion of a line element in the X-Y plane, in the Z direction, to create 4-noded SURF252 elements. *NSECT* indicates how many elements will be created. *SVAL* is the starting Z value, and *EVAL* is the ending Z value. *CS* must be 0.

CEXT

Circumferential extrusion (theta direction) around the global Y-axis. A 2-noded line element in the X-Y plane is extruded to create 4-noded SURF252 elements. *NSECT* indicates how many elements will be created. *SVAL* is the starting angle, and EVAL is the ending angle (in degrees). The angles are with respect to the global X-axis. *CS* must be 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(blank)

Cyclic reflection.

NSECT

Number of cyclic reflections to be done, or number of elements in the extrusion direction.

For planar reflection, this field must be 0 or blank.

For cyclic reflection, this field must be $\geq$ 1 or $\leq$ -1. Use a positive value if you want the sector angle to be computed automatically. Use a negative value if you want the sector angle to be computed manually. See Notes (p. 1634) for details.

CONDVALUE

Condensation key. Valid only when *Option* = COND.

ON

Activates condensation in the radiosity solver for all defined radiation symmetries/extrusions.

OFF

Deactivates condensation in the radiosity solver for all defined radiation symmetries/extrusions (default).

SVAL, EVAL

Starting and ending Z values (if *Axis* = ZEXT) or angle values (if *Axis* = CEXT) used for the extrusion. Not used for planar or cyclic reflection.

Notes

The **RSYMM (p. 1632)** command is used to define the plane of symmetry (POS) for planar reflection or the center of rotation (COR) for cyclic reflection. It can also be used to set parameters for a linear or circumferential extrusion. The input provided on this command is used to generate radiosity surface elements (SURF251/SURF252) when the **RSURF (p. 1631)** command is issued.

The **RSYMM (p. 1632)** command must be issued before **RSURF (p. 1631)**, and it may be issued multiple times to have more than one planar/cyclic reflection or extrusion. The **RSURF (p. 1631)** command processes **RSYMM (p. 1632)** commands in the order they are issued.

For planar reflection, you must define a local coordinate system (≥11) with its origin on the POS. One of its axes must be aligned so that it is normal to the plane. If possible, use the existing global coordinate system (0).

For cyclic reflection, you must define a local coordinate system (≥11) with its origin coincident with the COR. Reflections occur about the local Z-axis in the counterclockwise direction. You must align the Z-axis properly. If possible, use the existing global coordinate system (0).

For cyclic reflection, *NSECT* is used as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\Delta\theta = \frac{2\pi}{abs(NSECT)+1}$ if NSECT ≤ 1

$\Delta\theta = \theta_{max} - \theta_{min}$ if NSECT ≥ 1

where θ_{max} and θ_{min} are computed internally based on location of the RDSF (surface-to-surface radiation) flagged surfaces.

See Figure 9: Usage Example: Positive and Negative *NSECT* Values (p. 1635) for an example of *NSECT* usage.

For linear or circumferential extrusion (*Axis* = ZEXT or CEXT), you must ensure that the extruded area matches the area of the underlying element; otherwise, the results may not be correct. For example, in the case of PLANE55 elements with a planar depth = 10, use *Axis* = ZEXT and set *SVAL* and *EVAL* such that *EVAL* - *SVAL* = 10. Likewise, for axisymmetric PLANE55 elements, use *Axis* = CEXT and set *SVAL* and *EVAL* such that *EVAL* - *SVAL* = 360. You must also issue **V2DOPT (p. 2070)**,1 for the axisymmetric case. See Figure 10: Usage Example: Extrusions with *Axis* = ZEXT and CEXT (p. 1636) for extrusion examples.

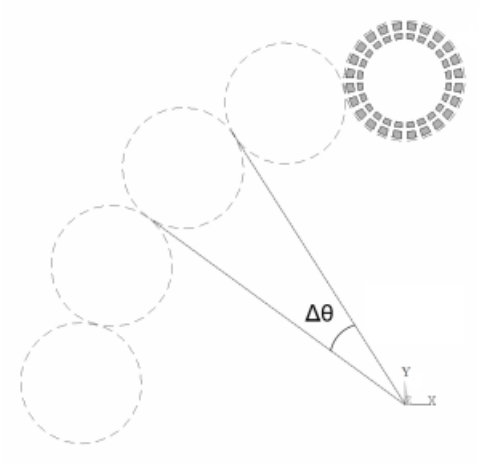
The *Axis*= ZEXT and CEXT options are not valid for SHELL131 and SHELL132 elements.

New surface elements generated by the **RSYMM (p. 1632)** command inherit the properties of the original elements.

For 2D axisymmetric models, **RSYMM (p. 1632)** can be used only for symmetrization in the YR plane. It cannot be used for the theta direction. Use **V2DOPT (p. 2070)** in that case.

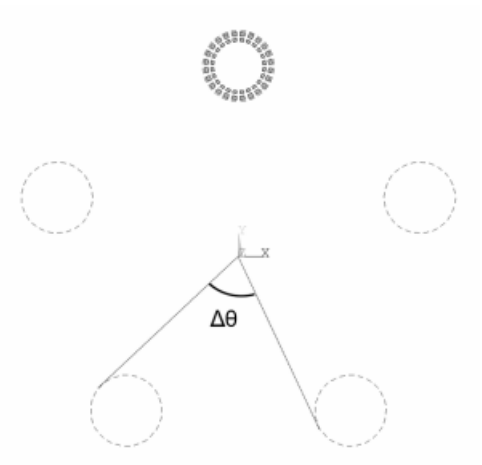
For 2D axisymmetric YR models, the newly-generated nodes can have only positive X coordinates.

Figure 9: Usage Example: Positive and Negative *NSECT* Values



RSYMM (p. 1632),,,4

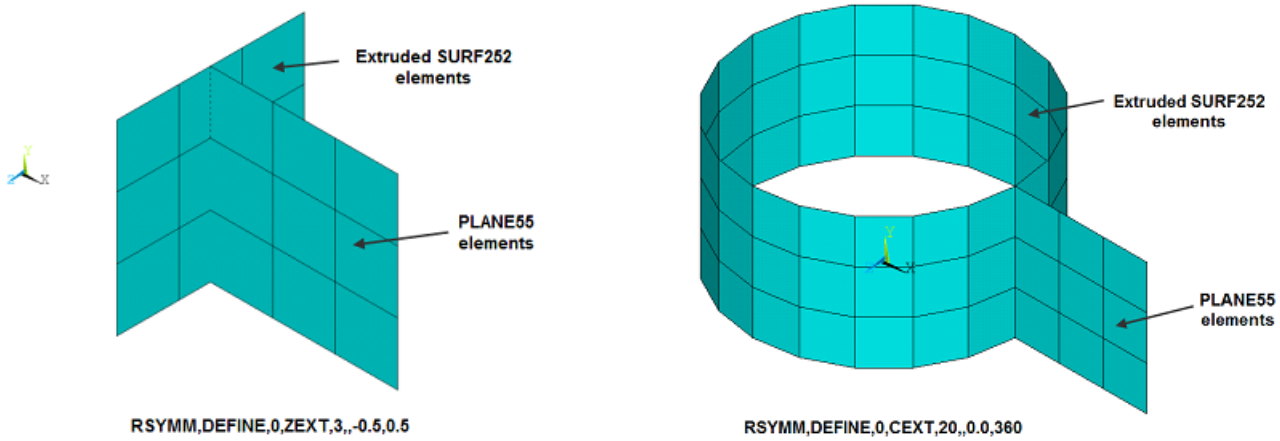
$\Delta\theta = \theta_{max} - \theta_{min}$



RSYMM (p. 1632),,,-4

$\Delta\theta = \frac{2\pi}{abs(-4)+1}$

Figure 10: Usage Example: Extrusions with *Axis* = ZEXT and CEXT

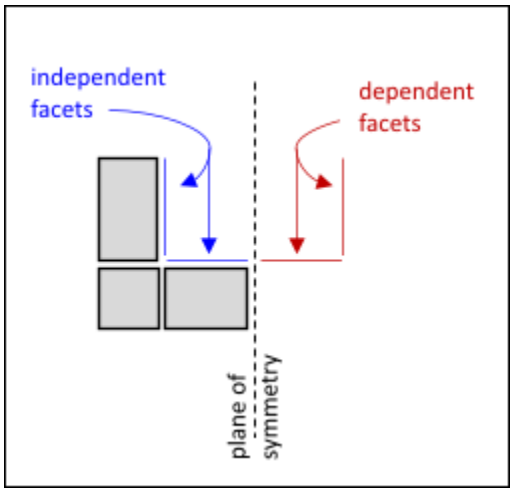


Considerations for View Factor Condensation

View Factor Condensation via the **VFCO (p. 2093)** command is the recommended method to improve solution efficiency for models with symmetry, which are defined with the **RSYMM (p. 1632)** command.

When the **RSYMM (p. 1632)** command is used, it implies that some radiation facets (SURF251 or SURF252) created by the **RSURF (p. 1631)** command will be a reflection of others. By definition, radiation facets with an underlying solid element are independent facets. Dependent facets are copies of the independent facets having the same dimensions but at different locations. The following figures illustrate solid elements (grey) and independent (blue) and dependent (red) facets for models with different types of symmetry. View factor condensation improves efficiency by condensing the view factor matrix to calculate view factors only for independent facets (View Factor Matrix for a Model with Symmetry) and simplifying the radiosity equations to solve only for the independent radiosity flux (Radiosity Equations Simplified for Models with Symmetry in the *Theory Reference*). For an illustrative example, see Example of a 3D Open Enclosure with Symmetry: Radiation Analysis with Condensed View Factor Calculation in the *Thermal Analysis Guide*.

Figure 11: Independent and Dependent Facets in a Model with Planar Symmetry Employing View Factor Condensation



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12: Independent and Dependent Facets in a Model with Cyclic Symmetry Employing View Factor Condensation

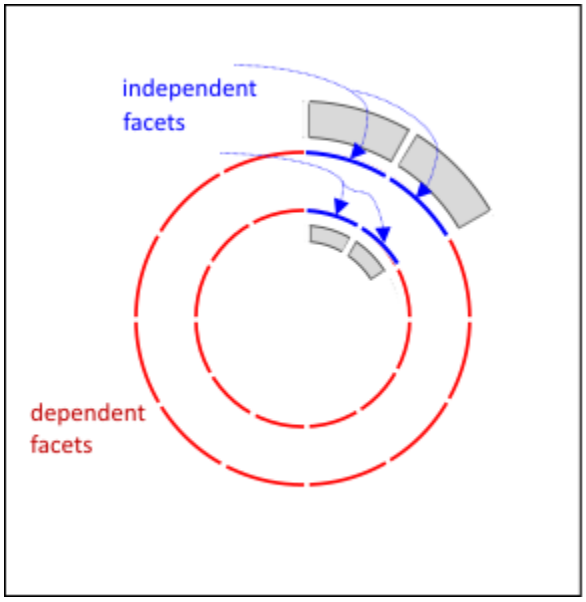
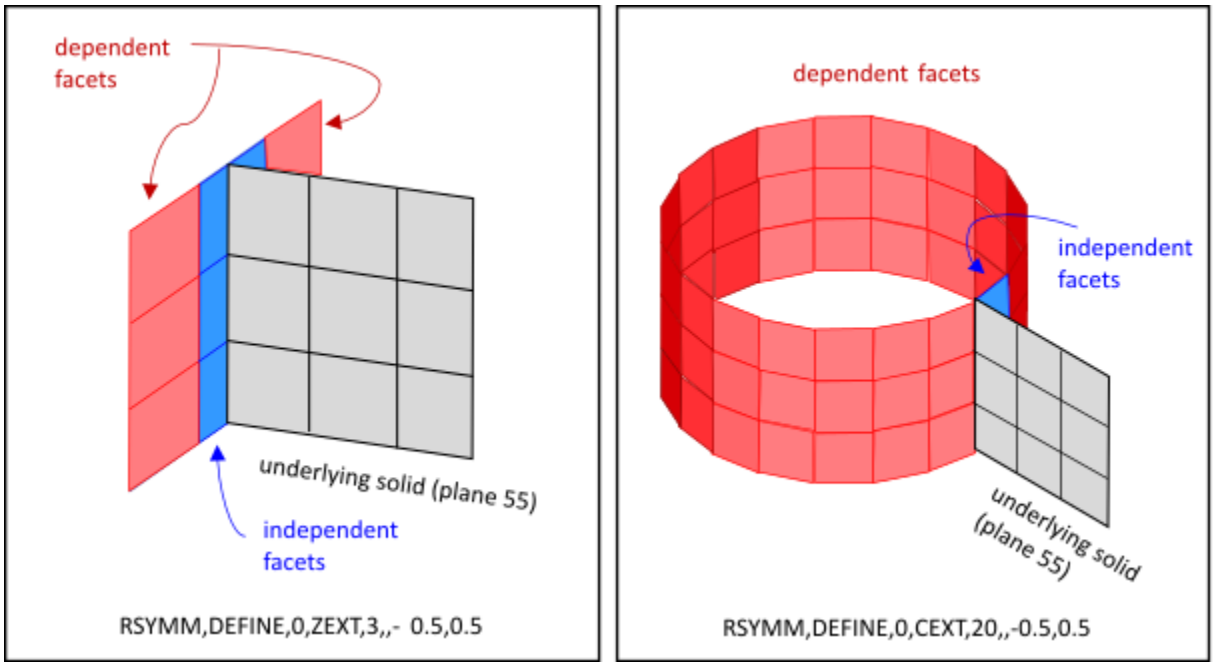


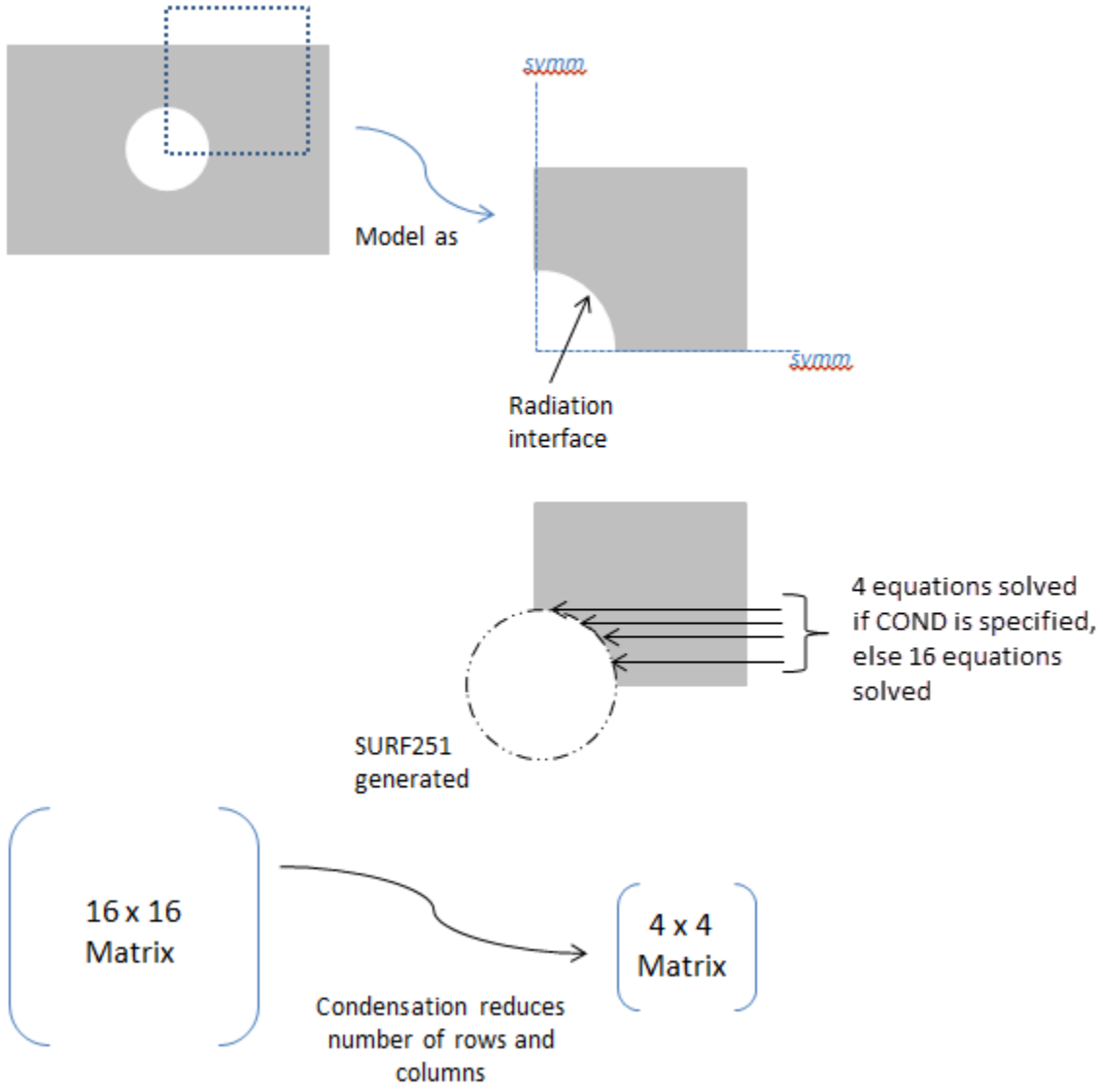
Figure 13: Independent and Dependent Facets for a Model Built by Extrusions Employing View Factor Condensation



Although it is not the recommended method, the following figure illustrates condensation via **RSYMM (p. 1632)**,COND. The efficiency gains by condensation via **RSYMM (p. 1632)**,COND are less than those obtained with view factor condensation via the **VFCO (p. 2093)** command, which reduces the view factor matrix in addition to simplifying the radiosity equations, as described in View Factor Matrix for a Model with Symmetry and Radiosity Equations Simplified for Models with Symmetry in the *Theory Reference*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14: Usage Example: *Option* = COND



Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Example Usage

2D Radiation Analysis Using the Radiosity Method with Decimation and Symmetry

3D Open Enclosure with Symmetry: Radiation Analysis with Condensed View Factor Calculation

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Clear Symmetry

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Condensation

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Cyclic Symmetry

Main Menu>Preprocessor>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Planar Symmetry

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Clear Symmetry

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Condensation

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Cyclic Symmetry

Main Menu>Solution>Radiation Opts>Advanced Solution Option>Radiation Symmetry Options>Planar Symmetry

---

RSYS, *KCN*

Activates a coordinate system for printout or display of element and nodal results.

POST1 (p. 51): Controls (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCN

The coordinate system to use for results output:

0

Global Cartesian coordinate system (default, except for spectrum analyses).

1

Global cylindrical coordinate system in Z.

2

Global spherical coordinate system.

5

Global cylindrical coordinate system in Y.

6

Global cylindrical coordinate system in X.

> 10

Any existing local coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLU

Solution coordinate systems.

LSYS

Layer coordinate system (default for spectrum analysis).

Notes

The **RSYS (p. 1639)** command activates a coordinate system for printing or displaying element results data such as stresses and heat fluxes, and nodal results data such as degrees of freedom and reactions.

Mechanical APDL rotates the results data to the specified coordinate system during printout, display, or element table operations (such as **PRNSOL (p. 1505)**, **PRESOL (p. 1481)**, **PLNSOL (p. 1425)**, and **ETABLE (p. 687)**).

You can define coordinate systems with various Mechanical APDL commands such as **LOCAL (p. 1059)**, **CS (p. 435)**, **CLOCAL (p. 352)**, and **CSKP (p. 438)**.

The **RSYS (p. 1639)** command has no effect on beam or pipe stresses, which Mechanical APDL displays (via **/ESHAPE (p. 665)**,1 and PowerGraphics) in the element coordinate system.

Element Results

Element results such as stresses and heat fluxes are in the element coordinate systems when *KCN* = SOLU. Nodal requests for element results (for example, **PRNSOL (p. 1505)**,S,COMP) average the element values at the common node; that is, the orientation of the node is not a factor in the output of element quantities.

For nearly all solid elements, the default element coordinate systems are parallel to the global Cartesian coordinate system.

For shell elements and the remaining solid elements, the default element coordinate system can differ from element to element.

For layered shell and layered solid elements, Mechanical APDL initially selects the element coordinate system when *KCN* = SOLU. You can then select the layer coordinate system via the **LAYER (p. 1015)** command.

Nodal Results

Nodal results such as degrees of freedom and reactions can be properly rotated only if the resulting component set is consistent with the degree-of-freedom set at the node. The degree-of-freedom set at a node is determined by the elements attached to the node.

> *Example:* If a node does not have a UZ degree of freedom during solution, any Z component resulting from a rotation does not print or display in POST1. Results at nodes with a single degree-of-freedom (UY only, for example) should therefore not be rotated; that is, they should be viewed only in the nodal coordinate system or a system parallel to the nodal system. (The default global Cartesian system cannot be parallel to the nodal system.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results at nodes with a 2D degree-of-freedom set (UX and UY, for example) should not be rotated out of the 2D plane.

Existing Local Coordinate Systems (*KCN* > 10)

When *KCN* > 10, and the specified system is subsequently redefined, reissue the **RSYS (p. 1639)** command for results to be rotated into the redefined system.

Solution Coordinate Systems (*KCN* = SOLU)

For element quantities, solution coordinate systems are the element coordinate system for each element. For nodal quantities, they are the nodal coordinate systems.

If an element or nodal coordinate system is not defined, Mechanical APDL uses the global Cartesian coordinate system.

If you issue a **LAYER (p. 1015)**,*N* command (where *N* refers to a layer number), the results appear in the layer coordinate system. (SOLU is the default for spectrum analyses.)

The default coordinate system for certain elements, notably shells, is not global Cartesian and is frequently not aligned at adjacent elements. Avoid setting *KCN* = SOLU with such elements, as it can make nodal averaging of component element results (such as SX, SY, SZ, SXY, SYZ, and SXZ) invalid.

When post-processing expanded nodal results in a cyclic symmetry analysis (**/CYCEXPAND (p. 454)**), use of **RSYS (p. 1639)**,SOLU is recommended so that the appropriate cyclic nodal coordinate system is used (see **CYCLIC (p. 462)** command). For any other coordinate system (for example, **RSYS (p. 1639)**,1), cyclic rotation is not carried out, and nodal results at all sectors are expressed in the specified coordinate system (*KCN*). See Result Coordinate System in the *Cyclic Symmetry Analysis Guide* for more details.

Layered Coordinate System (*KCN* = LSYS)

For layered shell and solid elements, the results appear in their respective layer coordinate systems. For a specific layer of interest, issue a **LAYER (p. 1015)**,*N* command (where *N* refers to a layer number).

If a model has both nonlayered and layered elements, you can use **RSYS (p. 1639)**,SOLU and **RSYS (p. 1639)**,LSYS simultaneously (with **RSYS (p. 1639)**,SOLU applicable to nonlayered elements and **RSYS (p. 1639)**,LSYS applicable to layered elements).

To reverse effects of the LSYS option, issue an **RSYS (p. 1639)**,0 command.

LSYS is the default for spectrum analysis.

PowerGraphics (/GRAPHICS (p. 866),POWER)

Mechanical APDL plots **PLVECT (p. 1445)** vector arrow displays (such temperature, velocity, and force) in the global Cartesian coordinate system (**RSYS (p. 1639)** = 0). Subsequent operations revert to your original coordinate system.

When using solution coordinate systems for results output (**RSYS (p. 1639)**,SOLU), the deformed or displaced shape in a POST1 contour display can be unexpected (although the contours are displayed in the expected colors). The program *does not* rotate displacement values (Ux,Uy,Uz) to global; instead, the displacements (stored locally) are added directly to the global coordinates (X,Y,Z). For example, if

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in PREP7 the nodes are rotated 90 degrees about the z axis and the global Uy displacements are relatively large, the Ux values will be large, causing the model to display a large deformation in the global X direction.

Large Deflection

If large deflection is active (**NLGEOM (p. 1231)**,ON), Mechanical APDL rotates the element component result directions by the amount of rigid body rotation.

Mechanical APDL displays the element component results in the initial global coordinate system for the following elements: SHELL181, SHELL281, ELBOW290, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLID272, SOLID273, SOLID285, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227.

All other element result transformations, therefore, are also relative to the initial global system.

Nodal degree-of-freedom results are based on the *initial* geometry.

For all other element types, component results displayed in the co-rotated coordinate system include the element rigid body rotation from the initial global coordinate system, and all other element result transformations are relative to the rotated global system.

Menu Paths

Main Menu>General Postproc>Options for Outp

Utility Menu>List>Results>Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

S Commands

SABS, *KEY*

Specifies absolute values for element table operations.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Absolute value key:

0

Use algebraic values in operations.

1

Use absolute values in operations.

Command Default

Use algebraic values.

Notes

Causes absolute values to be used in the **SADD (p. 1643)**, **SMULT (p. 1814)**, **SMAX (p. 1805)**, **SMIN (p. 1808)**, and **SSUM (p. 1858)** operations.

Menu Paths

Main Menu>General Postproc>Element Table>Abs Value Option

SADD, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*, *CONST*

Forms an element table item by adding two existing items.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab1

First labeled result item in operation.

Lab2

Second labeled result item in operation (may be blank).

FACT1

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

CONST

Constant value.

Notes

Forms a labeled result (see **ETABLE (p. 687)** command) for the selected elements by adding two existing labeled result items according to the operation:

$$LabR = (FACT1 \times Lab1) + (FACT2 \times Lab2) + CONST$$

May also be used to scale results for a single labeled result item. If absolute values are requested (**SABS (p. 1643)**,1), absolute values of *Lab1* and *Lab2* are used.

Menu Paths

Main Menu>General Postproc>Element Table>Add Items

SALLOW,*STRS1*,*STRS2*,*STRS3*,*STRS4*,*STRS5*,*STRS6*

Defines the allowable stress table for safety factor calculations.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

STRS1, STRS2, STRS3, . . . , STRS6

Input up to six allowable stresses corresponding to the temperature points (**TALLOW (p. 1897)**).

Notes

Defines the allowable stress table for safety factor calculations (**SFACT (p. 1743)**,**SFCALC (p. 1747)**). Use the **STAT (p. 1861)** command to list current allowable stress table. Repeat **SALLOW (p. 1644)** to zero table and redefine points (6 maximum).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Safety factor calculations are not supported by PowerGraphics. Both the **SALLOW (p. 1644)** and **TALLOW (p. 1897)** commands must be used with the Full Model Graphics display method active.

Menu Paths

Main Menu>General Postproc>Safety Factor>Allowable Strs>Constant

Main Menu>General Postproc>Safety Factor>Allowable Strs>Reset Stress

Main Menu>General Postproc>Safety Factor>Allowable Strs>Temp-depend

---

SAVE,*Fname*,*Ext*,--,*Slab*

Saves all current database information.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname.

Ext

Filename extension (eight-character maximum).

The extension defaults to DB if *Fname* is blank.

--

Unused field.

Slab

Mode for saving the database:

ALL

Save the model data, solution data and post data (element tables, etc.). This value is the default.

MODEL

Save the model data (solid model, finite element model, loadings, etc.) only.

SOLU

Save the model data and the solution data (nodal and element results).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Saves all current database information to a file (`File.db`). In interactive mode, an existing `File.db` is first written to a backup file (`File.dbb`). In batch mode, an existing `File.db` is replaced by the current database information with no backup. The command should be issued periodically to ensure a current file backup in case of a system "crash" or a "line drop." It may also be issued before a "doubtful" command so that if the result is not what was intended the database may be easily restored to the previous state. A save may be time consuming for large models. Repeated use of this command overwrites the previous data on the file (but a backup file is first written during an interactive run). When issued from within POST1, the nodal boundary conditions in the database (which were read from the results file) will overwrite the nodal boundary conditions existing on the database file.

Internal nodes may be created during solution (for example, via the mixed u-P formulation or generalized plane strain option for current-technology elements, the Lagrangian multiplier method for contact elements or the MPC184 elements, or the quadratic or cubic option of the BEAM188 and PIPE288 elements). It is sometimes necessary to save the internal nodes in the database for later operations, such as cutting boundary interpolations (**CBDOF (p. 285)**) for submodeling. To do so, issue the **SAVE (p. 1645)** command after the first **SOLVE (p. 1822)** command.

In general, saving after solving is always a good practice.

If radiosity surface elements (SURF251 or SURF252) are present in the model, a radiosity mapping data file, *Fname*`.RSM`, is also saved when the **SAVE (p. 1645)** command is issued. For more information, see Advanced Radiosity Options in the *Thermal Analysis Guide*.

This command is valid in any processor.

Menu Paths

Utility Menu>File>Save as

Utility Menu>File>Save as Jobname.db

SBCLIST

Lists solid model boundary conditions.

`SOLUTION` (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Lists all solid model boundary conditions for the selected solid model entities. See also **DKLIST (p. 545)**, **DLLIST (p. 551)**, **DALIST (p. 493)**, **FKLIST (p. 765)**, **SFLLIST (p. 1781)**, **SFALIST (p. 1745)**, **BFLLIST (p. 256)**, **BFALIST (p. 233)**, **BFVLIST (p. 264)**, and **BFKLIST (p. 251)** to list items separately.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>List>Loads>Solid Model Loads

SBCTRAN

Transfers solid model loads and boundary conditions to the FE model.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Causes a manual transfer of solid model loads and boundary conditions to the finite element model. Loads and boundary conditions on unselected keypoints, lines, areas, and volumes are not transferred. Boundary conditions and loads will not be transferred to unselected nodes or elements. The **SBCTRAN (p. 1647)** operation is also automatically done upon initiation of the solution calculations (**SOLVE (p. 1822)**).

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>All Solid Lds

Main Menu>Solution>Define Loads>Operate>Transfer to FE>All Solid Lds

---

*SCAL, *Name*, *Val1*, *VAL2*

Scales a vector or matrix by a constant or a vector.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name

Name used to identify the vector or matrix to be scaled. Must be specified.

Val1

When scaling a matrix or a vector by a scalar value, *Val1* is the real part of the constant to use (default = 1).

When scaling a matrix or a vector by a vector, *Val1* is the name of the vector used for the scaling operation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL2

The imaginary part of the constant to use (default = 0). This value is used only if the vector or matrix specified by *Name* is complex.

VAL2 is only valid for scaling by a constant. It is not used when scaling by a vector.

Notes

This command can be applied to vectors and matrices created by the ***VEC (p. 2085)**, ***DMAT (p. 551)** and ***SMAT (p. 1801)** commands.

Data types must be consistent between the vectors and matrices being scaled and the scaling vector (or constant value).

When scaling a matrix with a vector, the matrix must be square and the scaling vector must be the same size. See examples below.

Example Usage

Scaling a vector by a constant results in all vector values multiplied by the same constant value:

```
*SCAL,V,2             ! V[i] = V[i]*2
```

Scaling a matrix by a constant results in all matrix values multiplied by the same constant value:

```
*SCAL,M,2             ! M[i][j] = M[i][j]*2
```

The following command scales a complex values vector by a constant value of (2,1):

```
*SCAL,ZV,2,1          ! ZV[i] = ZV[i] * (2,1)
```

The following command scales a vector by another vector. All vector values are multiplied by the corresponding value in the scaling vector:

```
*SCAL,V,W  ! V[i] = V[i] * W[i]
```

The following command scales a matrix by a vector, resulting in all matrix values scaled using the formula below:

```
*SCAL,M,V  ! M[i][j] = M[i][j] * V[i] * V[j]
```

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SCOPT, *TempDepKey*,*MappingKey*

Specifies System Coupling options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TempDepKey

Temperature-dependent behavior key based on the convection coefficient:

YES

A negative convection coefficient, -*N*, is assumed to be a function of temperature and is determined from the HF property table for material *N* (**MP (p. 1160)** command). This is the default behavior.

NO

A negative convection coefficient, -*N*, is used as is in the convection calculation.

MappingKey

Controls whether midside nodes of higher-order elements are used for mapping on System Coupling interfaces:

YES

Both corner and midside nodes are used (default).

NO

Only corner nodes are used.

Notes

By default in the Mechanical APDL program, a negative convection coefficient value triggers temperature-dependent behavior. In some one-way CFD to Mechanical APDL thermal simulations, it is desirable to allow convection coefficients to be used as negative values. To do so, issue the command **SCOPT (p. 1649)**,NO.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SDELETE, *SFIRST*, *SLAST*, *SINC*, *KNOCLEAN*, *LCHK*

Deletes sections from the database.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SFIRST

First section ID to be deleted; defaults to first available section in the database.

SLAST

Last section ID to be deleted; defaults to last available section in the database.

SINC

Increment of the section ID; defaults to 1.

KNOCLEAN

Pretension element cleanup key (pretension sections only).

0

Perform cleanup of PRETS179 pretension elements (delete pretension elements and reconnect elements split during **PSMESH (p. 1543)**).

1

Do not perform cleanup.

LCHK

Specifies the level of element-associativity checking:

NOCHECK

No element-associativity check occurs. This option is the default.

WARN

When a section, material, or real constant is associated with an element, the program issues a message warning that the necessary entity has been deleted.

CHECK

The command terminates, and no section, material, or real constant is deleted if it is associated with an element.

Notes

Deletes one or more specified sections and their associated data from the Mechanical APDL database.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Sections>Delete Section

SE,*File*,--,--,*TOLER*,*nStartVN*

Defines a superelement.

PREP7 (p. 22): Superelements (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

File

The name (case sensitive) of the file containing the original superelement matrix created by the generation pass (*Sename*.sub). The default is the current Jobname.

--,--

Unused fields.

TOLER

Tolerance used to determine if use pass nodes are noncoincident with master nodes having the same node numbers. Defaults to 0.0001. Use pass nodes will always be replaced by master nodes of the same node number. However, if a use pass node is more than *TOLER* away from the corresponding master node, a warning is generated.

nStartVN

Node number to be assigned to the first virtual node created to store the generalized coordinates in a component mode synthesis analysis. See "Notes" (p. 1651) for more information.

Notes

Defines a superelement by reading in the superelement matrices and master nodes from the superelement matrix file. The matrix file (File.sub) must be available from the substructure generation pass. The proper element type (MATRIX50) must be active (**TYPE (p. 2036)**) for this command. A scratch file called File.sord showing the superelement names and their corresponding element numbers is also written.

nStartVN should be chosen so as to offset the virtual node numbers from the other node numbers used in the model. Otherwise, *nStartVN* is internally set by the program to fulfill that condition. The node number defined through *nStartVN* is considered only if applied on the first issued **SE (p. 1651)** command. *nStartVN* can also be defined during the generation pass using the **CMSOPT (p. 378)** command. If *nStartVN* is defined on both **CMSOPT (p. 378)** and **SE (p. 1651)** commands, the larger number prevails.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>From .SUB File

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SECCONTROL, *VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*, *VAL11*, *VAL12*, *VAL13*

Supplements or overrides default section properties.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1*, *VAL2*, *VAL3*, . . . , *VAL13

Values, such as the length of a side or the numbers of cells along the width, that describe the geometry of a section. See the "Notes" (p. 1652) section of this command description for details about these values for the various section types.

Notes

The **SECCONTROL (p. 1652)** command is divided into these operation types: Beams (p. 1652), Links (p. 1652), Pipes (p. 1653), Shells (p. 1653), and Reinforcings (p. 1654).

Values are associated with the most recently issued **SECTYPE (p. 1697)** command. The data required is determined by the section type and is different for each type.

SECCONTROL (p. 1652) overrides the program-calculated transverse-shear stiffness.

The command does not apply to thermal shell elements SHELL131 and SHELL132 or thermal solid elements SOLID278 and SOLID279.

Beams

Type: BEAM

Data to provide in the value fields (*VAL1* through *VAL4*):

TXZ – User transverse shear stiffness.

-- – Unused field.

TXY – User transverse shear stiffness.

ADDMAS – Added mass per unit length.

Links

Type: LINK

Data to provide in the value fields (*VAL1*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*):

ADDMAS – Added mass per unit length.

TENSKEY – Flag specifying tension and compression, tension only, or compression only (not valid for CABLE280:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 – Tension *and* compression (default).

1 – Tension only.

-1 – Compression only.

CV1, *CV2* – Damping coefficients.

CV3 – Compressive stiffness scaling factor (for CABLE280 only). The ratio between compressive stiffness and tensile stiffness. Default – 1.0e-5. Maximum – 1.0.

CV4 – Viscous regularization factor (for CABLE280 only). Default – 0.05. Maximum – 1.0.

Pipes

Type: PIPE

Data to provide in the value field (*VAL1*):

ADDMAS – Added mass per unit length. Use this value to account for extra hardware only.

Notes

Other masses are handled as follows:

- The mass of the internal fluid is accounted for by M_{int} on the **SECDATA (p. 1655)** command.
- The mass of the outer covering (insulation) is accounted for by M_{ins} on the **SECDATA (p. 1655)** command.
- The mass of the external fluid is accounted for by *MATOC* on the **OCDATA (p. 1307)** command.

Shells

Type: SHELL

Data to provide in the value fields (*VAL1* through *VAL8*):

E_{11} – User transverse-shear stiffness.

E_{22} – User transverse-shear stiffness.

E_{12} – User transverse-shear stiffness.

ADDMAS – Added mass-per-unit area.

HMEMSCF – Hourglass-control membrane-scale factor.

HBENSCF – Hourglass-control bending-scale factor.

DRLSTIF – Drill-stiffness scale factor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BENSTIF – Bending-stiffness scale factor (SHELL181 and SHELL281).

Reinforcing

Type: REINF

Data to provide in the value fields (*VAL1*, *VAL2*, *VAL3*):

TENSKEY – Specifies tension-and-compression, tension-only, or compression-only reinforcing behavior (valid for structural reinforcing analysis):

0 – Tension *and* compression (default).

1 – Tension only.

-1 – Compression only.

––––––––––––

REMBASE – Specifies how base-element material is handled:

0 – Retain base-element material in the space occupied by the reinforcing fibers (default).

1 – Remove base-element material in the space occupied by the reinforcing fibers.

––––––––––––

STSSTATE – Specifies the reinforcing stress state or heat flow:

For smeared reinforcing:

0 – Uniaxial-stress state (for structural reinforcing analysis) *or* uniaxial heat flow (for thermal reinforcing analysis). Only *kxx* is required. (Default.)

1 – Plane-stress state (for structural reinforcing analysis) *or* anisotropic heat flow (for thermal reinforcing analysis). Both kxx and kyy are specified.

2 – Plane-stress state with transverse shear stiffness. Valid for 3D smeared structural reinforcing analysis.

3 – Plane-stress state with transverse shear stiffness and bending stiffness. Valid for 3D smeared structural reinforcing analysis with solid base elements.

For discrete reinforcing:

0 – Uniaxial stiffness, or uniaxial heat flow for thermal reinforcing analysis. (Default.)

1 – Uniaxial, bending, and torsional stiffness with square cross section. Valid for 3D structural reinforcing analysis with solid base elements.

2 – Uniaxial, bending, and torsional stiffness with circular cross section. Valid for 3D structural reinforcing analysis with solid base elements.

Notes

REMBASE = 1 typically leads to more accurate models. (The base material must support 1D stress states.) For structural-reinforcing analysis, the base-element material consists of mass, stiffness, and body force. For thermal-reinforcing analysis, the base-element material consists of damping, conduc-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tion, and heat generation, and the base-element surface loads (convection and heat flux) are not subtracted. This option is not valid when the base-element material is anisotropic.

For smeared reinforcing with *STSSTATE* = 0, the equivalent thickness h of the smeared reinforcing layer is determined by h = *A* / *S*, where *A* is the cross-section area of a single fiber and *S* is the distance between two adjacent fibers. (See **SECDATA (p. 1655)**.)

STSSTATE = 1 to 3 is suitable for homogenous reinforcing layers (membrane) and applies to smeared reinforcing only (**SECTYPE (p. 1697)**,,REINF,SMEAR). For smeared reinforcing with *STSSTATE* = 1 to 3, discrete reinforcing with *STSSTATE* = 1 or 2, *TENSKEY* is ignored, and the default tension and compression behaviors apply to the reinforcing layers; also, the cross-section area input *A* is the thickness of the reinforcing layers and the distance input *S* is ignored. (See **SECDATA (p. 1655)** and REINF265 Structural/Thermal Input Data in the *Element Reference*.)

For smeared reinforcing with *STSSTATE* = 3 or discrete reinforcing with *STSSTATE* = 1 or 2, bending or torsional reinforcing stiffness may not be captured adequately when using reinforcing with overly refined high-order base tetrahedral elements (SOLID187 or degenerated SOLID186) and the stiffness ratio between reinforcing and base elements is excessive (> 100x).

Specified *TENSKEY*, *REMBASE* and *STSSTATE* values apply to *all* fibers defined in the current section.

For more information, see Reinforcing and Direct Element Embedding in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Sect Control

Main Menu>Preprocessor>Sections>Link>Add

Main Menu>Preprocessor>Sections>Link>Edit

Main Menu>Preprocessor>Sections>Pipe>Add

Main Menu>Preprocessor>Sections>Pipe>Edit

Main Menu>Preprocessor>Sections>Reinforcing>Sect Control

Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit

SECDATA,*VAL1*,*VAL2*,*VAL3*,*VAL4*,*VAL5*,*VAL6*,*VAL7*,*VAL8*,*VAL9*,*VAL10*,*VAL11*,*VAL12*

Describes the geometry of a section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VAL1, VAL2, VAL3, . . . , VAL12

Values, such as thickness or the length of a side or the numbers of cells along the width, that describe the geometry of a section. The terms *VAL1*, *VAL2*, etc. are specialized for each type of cross-section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The **SECDATA (p. 1655)** command defines the data describing the geometry of a section. The command is divided into these section types: Beams (p. 1656), Contact (p. 1659), General Axisymmetric (p. 1661), Joints (p. 1662), Links (p. 1663), Pipes (p. 1663), Pretension (p. 1664), Reinforcing (p. 1665), Shells (p. 1675), Supports (p. 1676), and Taper (p. 1677).

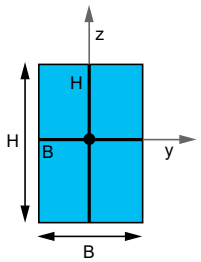
The data input on the **SECDATA (p. 1655)** command is interpreted based on the most recently issued **SECTYPE (p. 1697)** command. The data required is determined by the section type and subtype, and is different for each one.

Beams

Type: BEAM

Beam sections are referenced by BEAM188 and BEAM189 elements. Not all **SECOFFSET (p. 1685)** location values are valid for each subtype.

Type: BEAM, Subtype: RECT



Data to provide in the value fields:

B, *H*, *Nb*, *Nh*

where:

B = Width

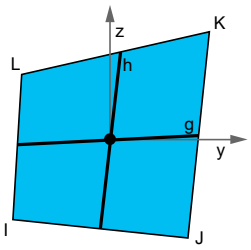
H = Height

Nb = Number of cells along width; default = 2

Nh = Number of cells along height; default = 2

*Nb*Nh*

Type: BEAM, Subtype: QUAD



Data to provide in the value fields:

yI, *zI*, *yJ*, *zJ*, *yK*, *zK*, *yL*, *zL*, *Ng*, *Nh*

yI, *zI*, *yJ*, *zJ*, *yK*, *zK*, *yL*, *zL* = Coordinate location of various points

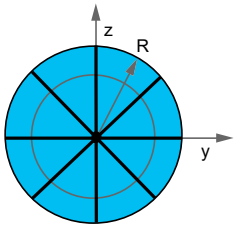
Ng = Number of cells along *g*; default = 2

Nh = Number of cells along *h*; default = 2

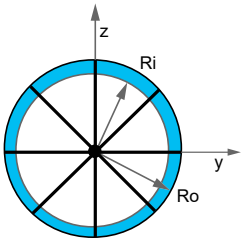
*Ng*Nh*

Degeneration to triangle is permitted by specifying the same coordinates for cells along an edge.

Type: BEAM, Subtype: CSOLID



Type: BEAM, Subtype: CTUBE



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data to provide in the value fields:

R, *N*, *T*

where:

R = Radius

N = Number of divisions around the circumference; 8 ≤ *N* ≤ 120 (where a greater value improves accuracy slightly); default = 8

T = Number of divisions through the radius; default = 2

Data to provide in the value fields:

Ri, *Ro*, *N*

where:

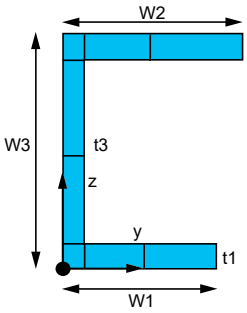
Ri = Inner radius of the tube

Ro = Outer radius of the tube

N = Number of cells around the circumference; N ≥ 8 (where a greater value improves accuracy slightly); default = 8

This subtype is similar to type PIPE (p. 1663). However, elements using PIPE account for internal or external pressures, whereas elements using CTUBE do not.

Type: BEAM, Subtype: CHAN



Data to provide in the value fields:

W1, *W2*, *W3*, *t1*, *t2*, *t3*

where:

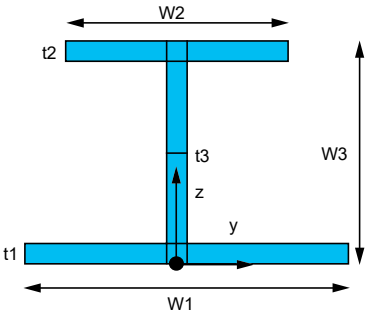
W1, *W2* = Lengths of the flanges

W3 = Overall depth

t1, *t2* = Flange thicknesses

t3 = Web thicknesses

Type: BEAM, Subtype: I



Data to provide in the value fields:

W1, *W2*, *W3*, *t1*, *t2*, *t3*

where:

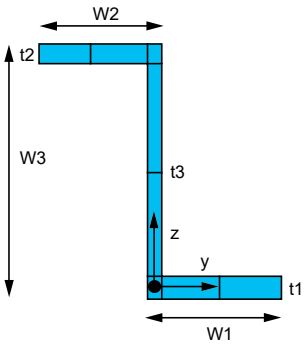
W1, *W2* = Width of the top and bottom flanges

W3 = Overall depth

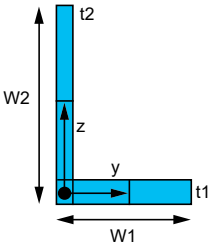
t1, *t2* = Flange thicknesses

t3 = Web thicknesses

Type: BEAM, Subtype: Z



Type: BEAM, Subtype: L



Data to provide in the value fields:

W1, *W2*, *t1*, *t2*

Data to provide in the value fields:

W1, *W2*, *W3*, *t1*, *t2*, *t3*

where:

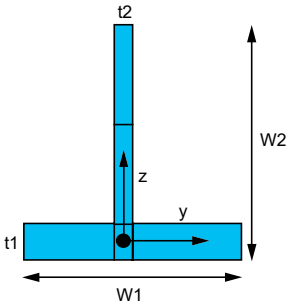
W1, *W2* = Flange lengths

W3 = Overall depth

t1, *t2* = Flange thicknesses

t3 = Stem thicknesses

Type: BEAM, Subtype: T



Data to provide in the value fields:

W1, *W2*, *t1*, *t2*

where:

W1 = Flange width

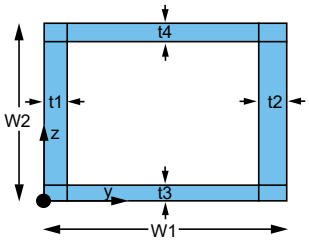
W2 = Overall depth

t1 = Flange thicknesses

t2 = Stem thicknesses

If *W2* is a negative value, the section will be flipped.

Type: BEAM, Subtype: HREC



Data to provide in the value fields:

W1, *W2*, *t1*, *t2*, *t3*, *t4*

where:

W1 = Outer width of the box

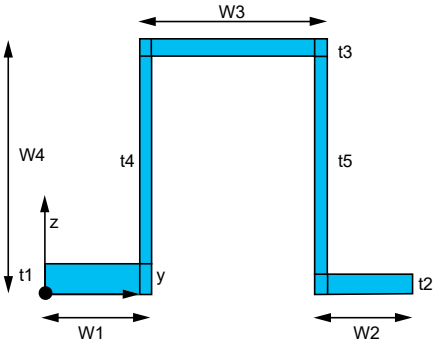
where:

W1, *W2* = Leg lengths

t1, *t2* = Leg thicknesses

If *W2* is a negative value, the section will be flipped.

Type: BEAM, Subtype: HATS



Data to provide in the value fields:

W1, *W2*, *W3*, *W4*, *t1*, *t2*, *t3*, *t4*, *t5*

where:

W1, *W2* = Width of the brim

W3 = Width of the top of the hat

W4 = Overall depth

t1, *t2* = Thickness of the brim

t3 = Thickness of the top of the hat

t4, *t5* = Web thicknesses

Type: BEAM, Subtype: ASEC

Arbitrary: User-supplied integrated section properties instead of basic geometry data.

Data to provide in the value fields:

A, *Iyy*, *Iyz*, *Izz*, *Iw*, *J*, *CGy*, *CGz*, *SHy*, *SHz*, *TKz*, *TKy*, *TSxz*, *TSxy*

where:

A = Area of section

Iyy = Moment of inertia about the y axis

Iyz = Product of inertia

Izz = Moment of inertia about the z axis

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

W2 = Outer height of the box

t1, *t2*, *t3*, *t4* = Wall thickness

Iw = Warping constant

J = Torsional constant

CGy = y coordinate of centroid

CGz = z coordinate of centroid

SHy = y coordinate of shear center

SHz = z coordinate of shear center

TKz = Thickness along Z axis (maximum height)

TKy = Thickness along Y axis (maximum width)

TSxz = Shear-correction factor for component xz

TSxy = Shear-correction factor for component xy

SECPLOT (p. 1688) cannot display an ASEC plot.

Type: BEAM, Subtype: MESH

User-defined mesh.

Data required is created by the **SECWRITE (p. 1703)** command and is read in by the **SECREAD (p. 1693)** command.

When a user mesh is input via **SECREAD (p. 1693)**, the program calculates the area, second moments of inertia, centroid, torsion constant, warping rigidity, and shear center.

If you redefine a material for a composite cross-section after creating the section, reissue the **SECREAD (p. 1693)** command.

CONTACT

Type: CONTACT

Geometry Correction

Contact sections for geometry correction (*Subtype* = CIRCLE, SPHERE, or CYLINDER) are referenced by the following elements: TARGE169, TARGE170, CONTA172, and CONTA174. This geometry correction applies to cases where the original meshes of contact elements or target elements are located on a portion of a circular, spherical, or revolute surface.

Type: CONTACT, Subtype: CIRCLE

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data to provide in the value fields for *Subtype* = CIRCLE:

X0, *Y0* (circle center location in Global Cartesian coordinates - XY plane)

Type: CONTACT, Subtype: SPHERE

Data to provide in the value fields for *Subtype* = SPHERE:

X0, *Y0*, *Z0* (sphere center location in Global Cartesian coordinates)

Type: CONTACT, Subtype: CYLINDER

Data to provide in the value fields for *Subtype* = CYLINDER:

X1, *Y1*, *Z1*, *X2*, *Y2*, *Z2* (two ends of cylindrical axis in Global Cartesian coordinates)

User-Defined Contact Surface Normal

The contact section for a user-defined contact surface normal (*Subtype* = NORMAL) is referenced by the following elements: CONTA172, CONTA174, and CONTA175. This geometry correction is used to define a shift direction for interference fit solutions.

Type: CONTACT, Subtype: NORMAL

Data to provide in the value fields for *Subtype* = NORMAL:

CSYS, *NX*, *NY*, *NZ*

where:

CSYS = Local coordinate system number (defaults to global Cartesian).

NX, *NY*, *NZ* = Direction cosines with respect to *CSYS*.

Radius values associated with contact or target elements

The radius contact section (*Subtype* = RADIUS) is referenced by contact or target elements in a general contact definition under the following circumstances:

- **Equivalent 3D contact radius for beam-to-beam contact** - The contact section for a user-defined equivalent contact radius (*Subtype* = RADIUS) is referenced by the element type CONTA177 within a general contact definition. 3D beam-to-beam contact (or edge-to-edge contact) modeled by this line contact element assumes that its surface is a cylindrical surface.
- **Radius (or radii) of rigid target segments** - The contact section for rigid target segment radii is referenced by target elements TARGE169 (circle segment type) and TARGE170 (line, parabola, cylinder, sphere, or cone segment type) in a general contact definition.

Type: CONTACT, Subtype: RADIUS

Data to provide in the value fields for *Subtype* = RADIUS if the section is used as an equivalent contact radius for 3D beam-to-beam contact:

VAL1 = Equivalent radius - outer radius

VAL2 = Equivalent radius - inner radius (internal beam-to-beam contact)

VAL3: Set to 1 for internal beam-to-beam contact. Defaults to external beam-to-beam contact.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data to provide in the value fields for *Subtype* = RADIUS if the section is used for 2D or 3D rigid target segments:

VAL1 = First radius of the target segment (used for circle, line, parabola, cylinder, sphere, and cone segment types)

VAL2 = Second radius of the target segment (used only for the cone segment type)

Simplified Bolt Thread Modeling

The contact section for bolt-thread modeling (*Subtype* = BOLT) is referenced by the following elements: CONTA172, CONTA174, and CONTA175. It applies to cases where the original meshes of contact elements are located on a portion of a bolt-thread surface. This feature allows you to include the behavior of bolt threads without having to add the geometric detail of the threads. Calculations are performed internally to approximate the behavior of the bolt-thread connections.

Type: CONTACT, Subtype: BOLT

Data to provide in the value fields for *Subtype* = BOLT:

D_m , *P*, *ALPHA*, *N*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

where:

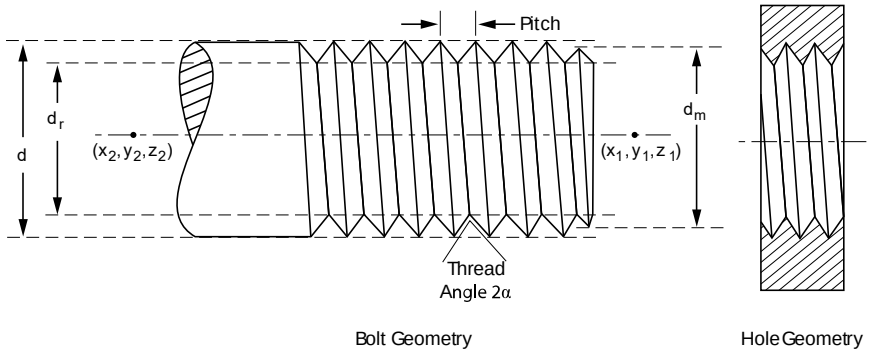
D_m = Pitch diameter, d_m.

P = Pitch distance, p.

ALPHA = Half-thread angle, α (defaults to 30 degrees).

N = Number of starts (defaults to 1).

X1, *Y1*, *Z1*, *X2*, *Y2*, *Z2* = Two end points of the bolt axis in global Cartesian coordinates.



General Axisymmetric

Type: AXIS

General axisymmetric sections are referenced by the SURF159, SOLID272, and SOLID273 elements. Use this command to locate the axisymmetric axis.

Data to provide in the value fields:

Pattern 1 (two points):

1, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2* are global Cartesian coordinates.

Pattern 2 (coordinate system number plus axis [1 = x, 2 = y, 3 = z]):

2, *csys*, *axis*

where *csys* is a Cartesian coordinate system.

Pattern 3 (origin plus direction):

3, *X0*, *Y0*, *Z0*, *xdir*, *ydir*, *zdir*

where *X0*, *Y0*, *Z0 are global Cartesian coordinates and xdir*, *ydir*, and *zdir* are direction cosines.

Joints

Type: JOINT

Joint sections are referenced by MPC184 joint elements.

Data to provide in the value fields:

length1, *length2*, *length3*, *angle1*, *angle2*, *angle3*

where:

length1-3 = Reference lengths used in the constitutive calculations.

angle1-3 = Reference angles used in the constitutive calculations.

The following table shows the lengths and angles to be specified for different kinds of joints.

Joint	Subtype	Reference lengths and angles
Revolute	REVO	angle1 for x-axis revolute joint angle3 for z-axis revolute joint
Universal	UNIV	angle1 and angle3
Slot	SLOT	length1
Point-in-plane	PINP	length2 and length3,
Revolute Translational	PRIS	length1
Cylindrical	CYLI	length1 and angle1 for x-axis cylindrical joint length3 and angle3 for z-axis cylindrical joint
Planar	PLAN	length2, length3, and angle1 for x-axis planar joint length1, length2, and angle3 for z-axis planar joint
Weld	WELD	(not used)
Orient	ORIE	(not used)
Spherical	SPHE	angle1, angle2, and angle3 (not used in friction calculations)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Joint	Subtype	Reference lengths and angles
General	GENE	length1, length2, length3, angle1, angle2, angle3 -- Depends upon which "unconstrained" relative degrees of freedom are available.
Screw	SCRE	length3 and angle3
Spotweld	SPWE	(not used)

The reference length and angle specifications correspond to the free relative degrees of freedom in a joint element for which constitutive calculations are performed. These values are used when stiffness and/or damping are specified for the joint elements.

If the reference lengths and angles are not specified, they are calculated from the default or starting configuration for the element.

See MPC184 or the individual joint element descriptions for more information on joint element constitutive calculations.

Links

Type: LINK

Link sections are referenced by the LINK33, LINK180, LINK228, and CABLE280 elements.

Data to provide in the value fields:

VAL1 = Area

Pipes

Type: PIPE

Pipe sections are referenced by the PIPE288, PIPE289, and ELBOW290 elements.

Data to provide in the value fields:

D_o, T_w, N_c, S_s, N_t, M_{int}, M_{ins}, T_{ins}

where:

D_o = Outside diameter of pipe. Use a constant value for a circular pipe and an array for a noncircular pipe. (Noncircular pipe sections are referenced by the ELBOW290 element only. See Defining a Noncircular Pipe in the *Structural Analysis Guide*.)

T_w = Wall thickness. Default = D_o / 2, or "solid" pipe. ("Solid" pipe is not applicable to ELBOW290; for that element, a thickness less than D_o / 4 is recommended.)

N_c = Number of cells around the circumference ($8 \le N_c \le 120$, where a greater value improves accuracy slightly; default = 8).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

S_s = Section number of the shell representing the pipe wall. Valid with ELBOW290 only. (Total thickness of the section is scaled to T_w. The program considers the innermost layer inside of the pipe to be the first layer.)

N_t = Number of cells through the pipe wall. Valid values are 1 (default), 3, 5, 7, and 9. Cells are graded such that they are thinner on the inner and outer surfaces. Valid with PIPE288 and PIPE289 only.

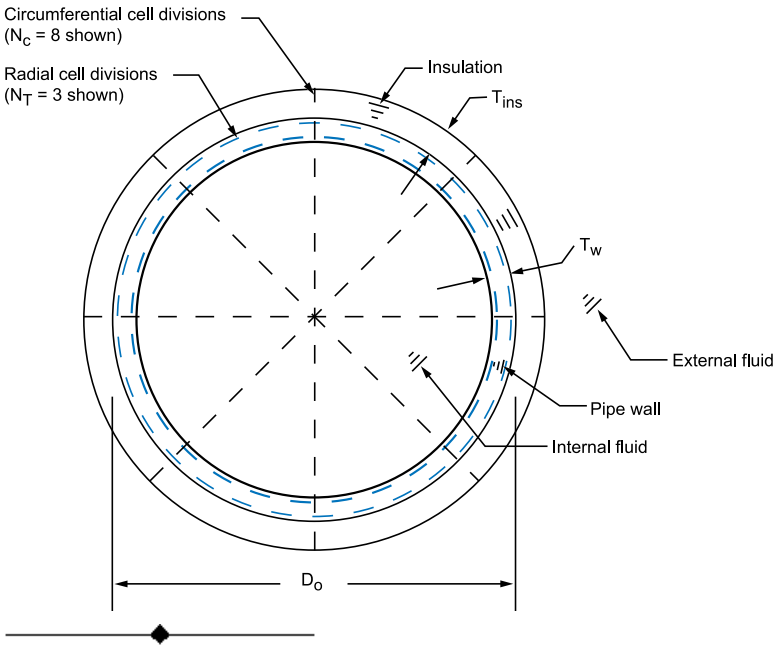
M_{int} = Material number of fluid inside of the pipe. The default value is 0 (no fluid). This value is used to input the density of the internal fluid. The fluid inside the pipe element is ignored unless the free surface in a global X-Y plane is added as face 3 (**SFE (p. 1761)**) and is high enough to include at least one end node of the element.

M_{ins} = Material number of material external to the pipe (such as insulation or armoring). The default value is 0 (no external material). This value is used to input the density of the external material. (External material adds mass and increases hydraulic diameter, but does not add to stiffness.)

T_{ins} = Thickness of material external to the pipe, such as insulation. The default value is 0 (no external material).

The accuracy of the ovalization value (OVAL) output by ELBOW290 (Structural Elbow form only) improves as the specified number of cells around the circumference (N_c) is increased.

External material (M_{ins}) adds mass and increases hydraulic diameter, but does not add to stiffness.



Pretension

Type: PRETENSION

Pretension sections are referenced by the PRETS179 element.

Data to provide in the value fields:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

node, *nx*, *ny*, *nz*

where:

node = Pretension node number.

nx = Orientation in global Cartesian x direction.

ny = Orientation in global Cartesian y direction.

nz = Orientation in global Cartesian z direction.

The following usage is typical:

```
SECTYPE, 1, PRETENSION
SECDATA, 13184, 0.000, 0.000, 1.000
SECMODIF, 1, NAME, example
SLOAD, 1, PL01, TINY, FORC, 100.00 , 1, 2
```

The *PRETENSION* section options of **SECTYPE (p. 1697)** and **SECDATA (p. 1655)** are documented mainly to aid in the understanding of data written by **CDWRITE (p. 293)**. Ansys, Inc. recommends that you generate pretension sections using **PSMESH (p. 1543)**.

Reinforcing

Type: REINF

Each **SECDATA (p. 1655)** command defines the material, geometry, and orientation (if Subtype = SMEAR) of one reinforcing member (discrete fiber or smeared surface) in the section. The reinforcing section can be referenced by reinforcing elements (REINF263, REINF264, and REINF265), or MESH200 elements when used for temporarily representing reinforcing members. Only one **SECDATA (p. 1655)** command is allowed per section when referenced by MESH200 elements. For more information, see Reinforcing and Direct Element Embedding in the *Structural Analysis Guide*.

- -

Type: REINF, Subtype: DISCRETE

Defines discrete reinforcing fibers with arbitrary orientations. For the MESH input pattern, reinforcing section data is referenced by MESH200 elements. For other patterns, issue separate **SECDATA (p. 1655)** commands to define each reinforcing fiber.

Data to provide in the value fields:

MAT, A, PATT, V1, V2, V3, V4, V5

MAT = Material ID for the fiber. (See REINF264 for valid material models.) When the reinforcing section is referenced by a MESH200 element, the default is the MESH200 element material ID (**MAT (p. 1111)**). When the section is referenced by reinforcing elements, the material ID is required for all fibers, and no default for this value is available.

A = Cross-section area of the reinforcing fiber.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PATT = Input pattern code (described below) indicating how the location of this fiber is defined. Available input patterns are MESH (when the section is referenced by a MESH200 element), and LAYN, EDGO, and BEAM (when the section is referenced by a reinforcing element).

V1, *V2*, *V3*, *V4*, *V5* = Values to define the location of the reinforcing fiber (depending on the *PATT* pattern code used), as shown:

***PATT* : MESH**

Description: The locations of reinforcing fibers are defined directly via MESH200 element connectivity.

Required input: None.

***PATT* : LAYN**

Description: The discrete reinforcing fiber is placed in the middle of a layer in a layered base element. The orientation of the fiber within the layer is adjustable via offsets with respect to a specified element edge.

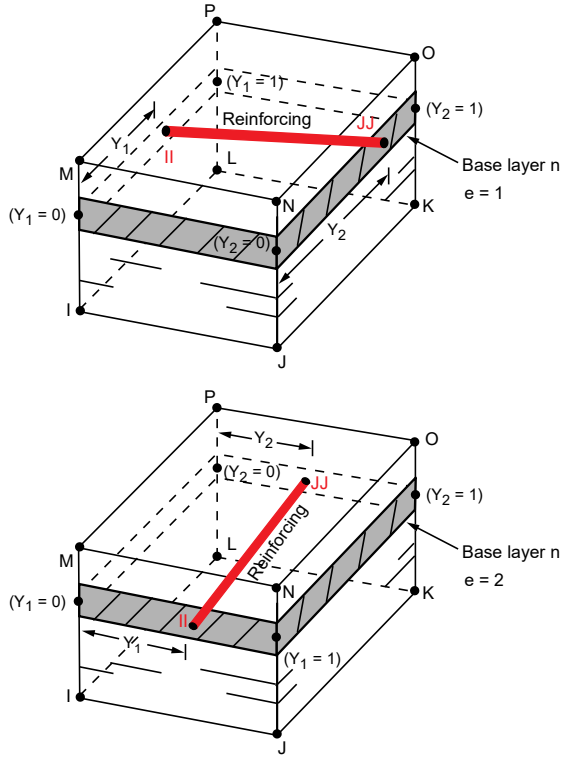
Required input:

V1 (or *N*) -- The number of the layer in the base element on which to apply the reinforcing fiber. The default value is 1.

V2 (or *e*) -- The number to indicate the element edge to which the offsets are measured. The default value is 1.

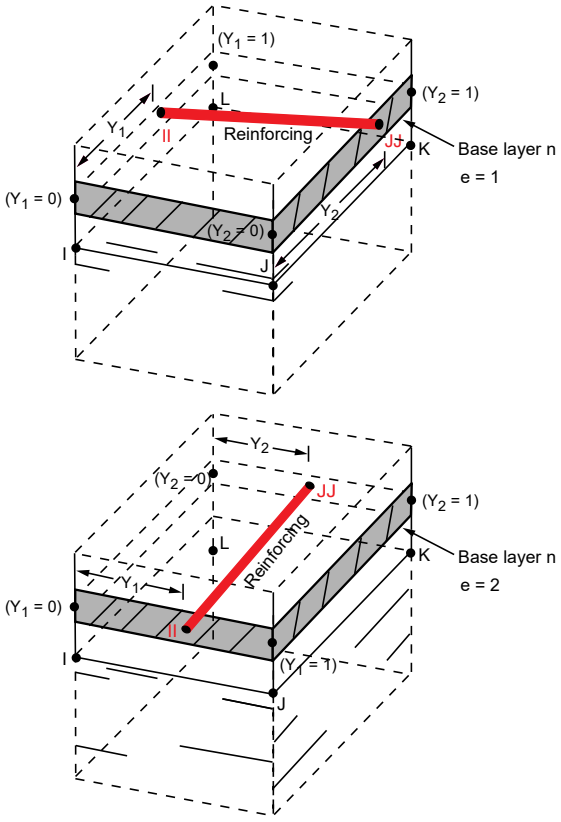
V3 and *V4* (or *Y1* and *Y2*) -- The normalized distances from the fiber to the two ends of the specified element edge. Valid values for *Y1* and *Y2* are 0.0 through 1.0. The default value of *Y1* is 0.5. The default value of *Y2* is *Y1*.

When applied to 8-node or 20-node layered solid elements:



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When applied to 4-node or 8-node layered shell elements:



***PATT* : EDGO**

Description: The orientation of the discrete reinforcing fiber is similar to one of the specified element edges. The fiber orientation can be further adjusted via offsets with respect to the specified element edge.

Required input:

V1 (or *0*) -- The number to indicate the element edge to which the offsets are measured. The default value is 1.

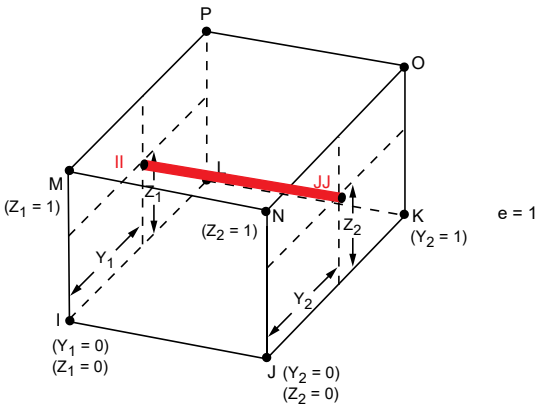
V2 and *V3* (or *Y1* and *Z1*) -- The normalized distances from the fiber to the first end of the specified element edge. Valid values for *Y1* and *Z1* are 0.0 through 1.0. The default value for *Y1* and *Z1* is 0.5.

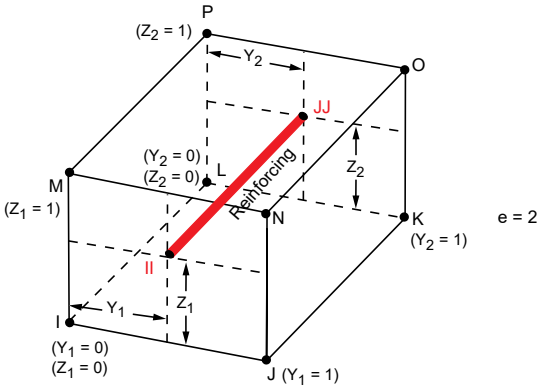
V4 and *V5* (or *Y2* and *Z2*) – The normalized distances from the fiber to the second end of the specified element edge. Value values for *Y2* and *Z2* are 0.0 through 1.0. The default value for *Y2* is *Y1*, and the default value for *Z2* is *Z1*.

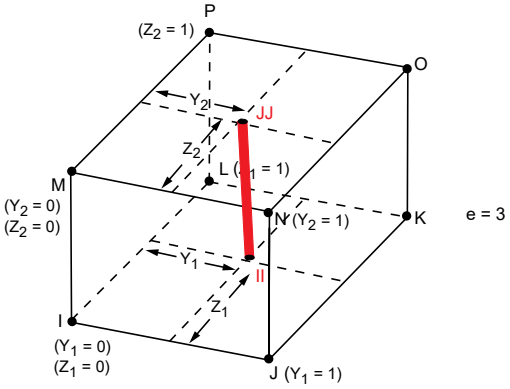
If the base element is a beam or link, the program ignores values *V2* through *V5* and instead places the reinforcing in the center of the beam or link.

When applied to 8-node or 20-node solid elements:

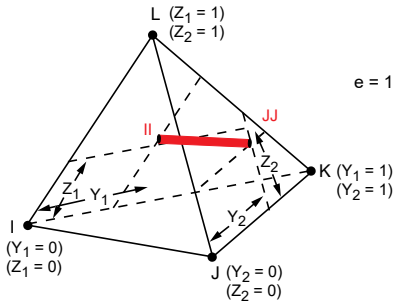
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



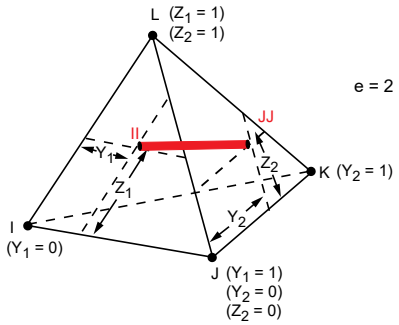


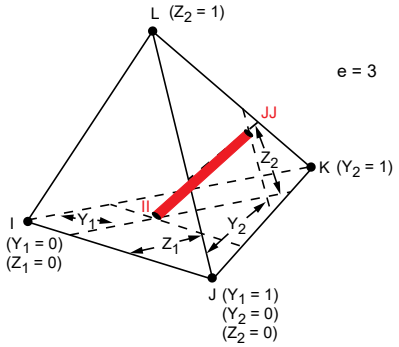


When applied to tetrahedral elements:

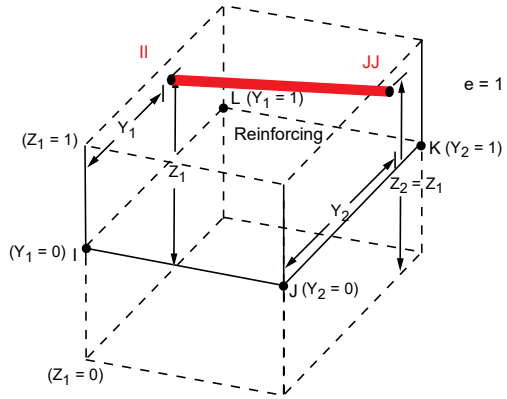


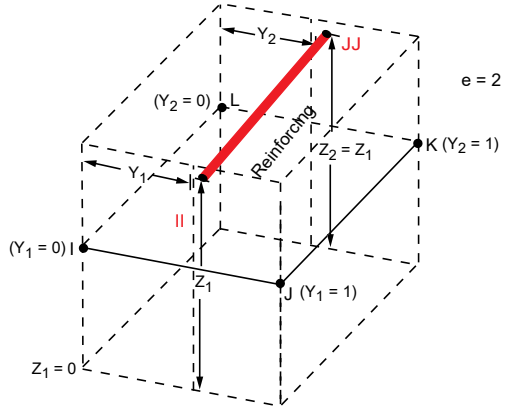
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





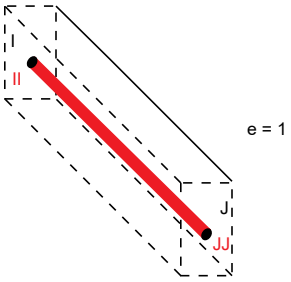
When applied to 3D shell elements:





When applied to beam or link elements:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



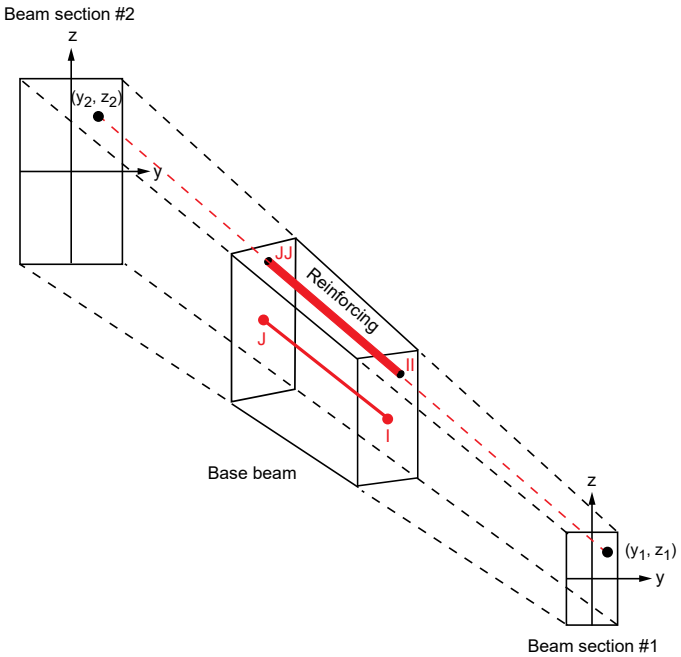
***PATT* : BEAM**

Description: Use this specialized input pattern for defining reinforcing in regular constant and tapered beams.

Required input:

V1 and *V2* (or *Y1* and *Z1*) -- Y and Z offsets with respect to the section origin in the first beam section referenced by the base beam element. The default value for Y1 and Z1 is 0.0.

V3 and *V4* (or *Y2* and *Z2*) -- Y and Z offsets with respect to the section origin in the second beam section referenced by the base beam element. The default value for *Y2* is *Y1*, and the default value for *Z2* is *Z1*. (Because *V3* and *V4* values apply only to tapered beams, the program ignores them if the base beam has a constant section.)



- -

Type: REINF, Subtype: SMEAR

Suitable for layers of reinforcing fibers with uniform cross-section area and spacing. Each **SECDATA (p. 1655)** command defines the one reinforcing layer in the section. When referenced by a MESH200 element, only one **SECDATA (p. 1655)** command per section is allowed. When referenced by reinforcing elements (REINF263 and REINF265), this limitation does not apply.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data to provide in the value fields:

MAT, A, S, KCN, THETA, PATT, V1, V2, V3, V4, V5

where:

MAT = Material ID for layer. (See REINF263 or REINF265 for available material models.) When the section is referenced by a MESH200 element, the default is the MESH200 element material ID (**MAT (p. 1111)**). When the section is referenced by reinforcing elements, the material ID is required for all fibers, and no default for this value is available.

A = Cross-section area of a single reinforcing fiber (*or* the thickness of the reinforcing layer for homogeneous reinforcing membranes).

S = Distance between two adjacent reinforcing fibers (ignored for homogeneous reinforcing membranes).

Note: If the section is used to model the reinforcing layers with a uniaxial stress state (**SECCONTROL (p. 1652)**,,,0), the equivalent thickness h of the reinforcing layer is determined by h = *A* / *S*, where *A* is the cross-section area of a single fiber and *S* is the distance between two adjacent fibers. If the section is used to model homogeneous reinforcing membranes (**SECCONTROL (p. 1652)**,,,1), the cross-section area input *A* is the thickness of the reinforcing layer and the distance input *S* is ignored.

KCN = Local coordinate system reference number for this layer. (See **LOCAL (p. 1059)** for more information.) When the section is referenced by a MESH200 element, the default *KCN* value is the MESH200 element coordinate system ID (**ESYS (p. 685)**). For the 2D smeared reinforcing element REINF263, *KCN* input is not required. When *KCN* is not specified, the program uses a default layer coordinate system (described in REINF263 and REINF265).

THETA = Angle (in degrees) of the final layer coordinate system with respect to the default layer system or the layer system specified in the *KCN* field. This value is ignored for REINF263 when that element is embedded in 2D plane strain or plane stress base elements.

PATT = Input pattern code (described below) indicating how the location of this fiber is defined. Available input patterns are MESH (when the section is referenced by a MESH200 element), and LAYN, EDGO, and BEAM (when the section is referenced by a reinforcing element).

V1, V2, V3, V4, V5 = Values to define the location of the reinforcing layer, as shown:

***PATT* : MESH**

Description: The locations of reinforcing fibers are defined directly via MESH200 element connectivity.

Required input: None.

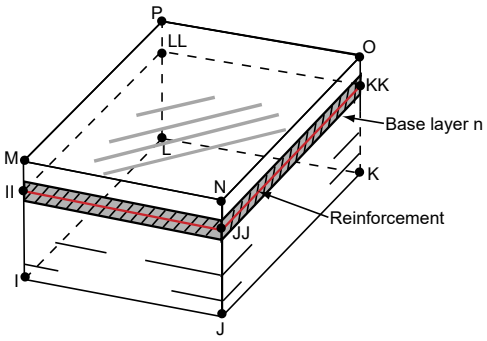
***PATT* : LAYN**

Description: The smeared reinforcing layer is placed in the middle of a layer in a layered base element.

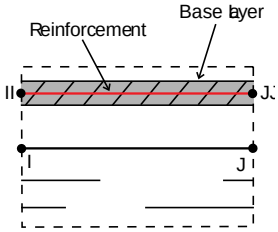
Required input: *V1* (or *n*) -- The number of the layer in the base element on which to apply the reinforcing layer. The default value is 1.

When applied to layered solid elements:

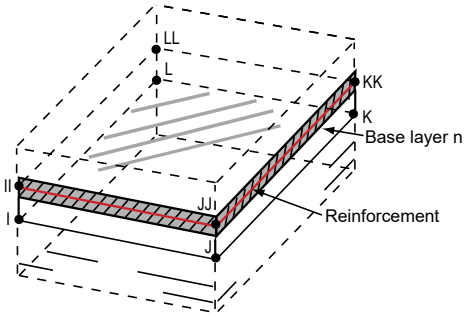
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When applied to 2D axisymmetric shells:



When applied to 3D layered shells:



***PATT* : EDGO**

Description: This pattern applies only to 2D smeared reinforcing element REINF263. The smeared reinforcing layer is represented by a line in 2D. The orientation of the 2D smeared reinforcing layer is similar to one of the specified element edges. The fiber orientation can be further adjusted via offsets with respect to the specified element edge.

Required input:

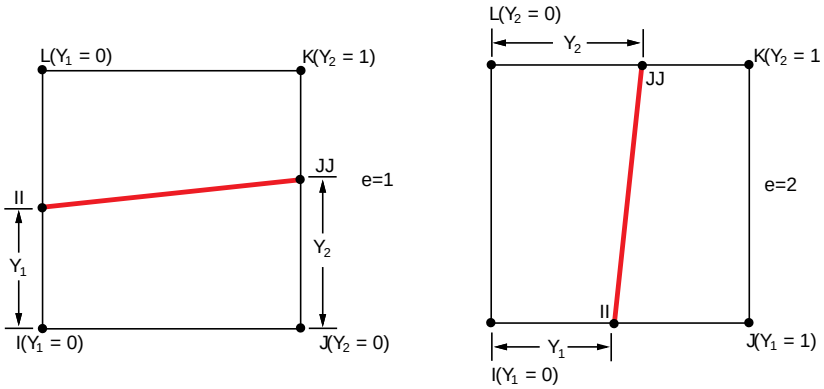
V1 (or *0*) -- The number to indicate the element edge to which the offsets are measured. The default value is 1.

V2 (or *Y1*) -- The normalized distances from the reinforcing layer to the first end of the specified element edge. Valid values for Y1 are 0.0 through 1.0. The default value for *Y1* is 0.5. *V3* (or *Z1*) input is ignored.

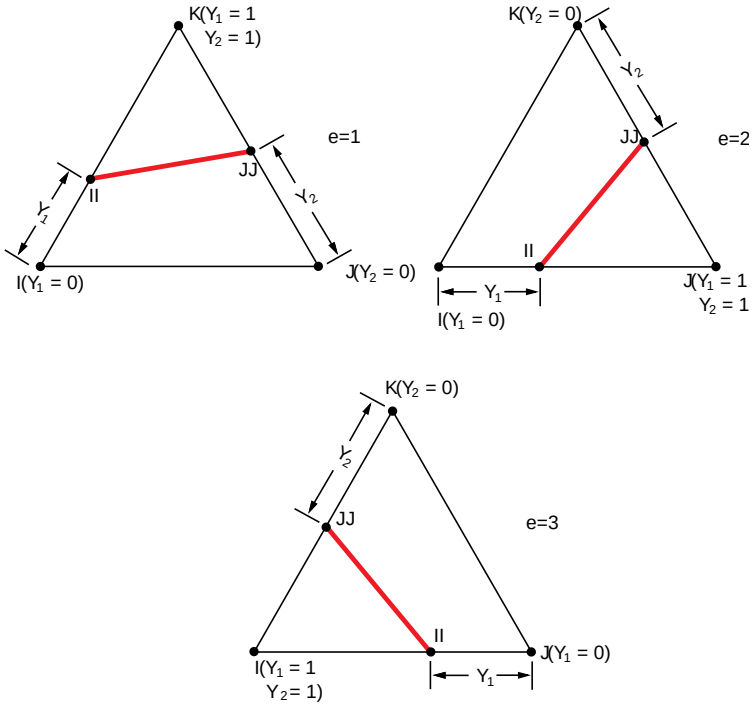
V4 (or *Y2*) -- The normalized distances from the reinforcing layer to the second end of the specified element edge. Valid value values for *Y2* are 0.0 through 1.0. The default value for *Y2* is *Y1*. *V4* (or *Y2*) is ignored for axisymmetric shell elements. *V5* (or *Z2* `) input is ignored.

When applied to 2D 4-node or 8-node solid elements:

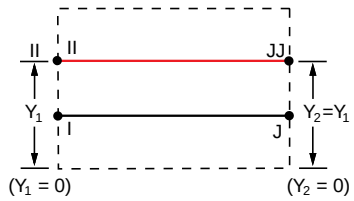
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When applied to 2D 3-node or 6-node triangular solid elements:



When applied to 2-node or 3-node axisymmetric shell elements:



***PATT* : ELEF**

Description: The smeared reinforcing layer is oriented parallel to one of three adjacent element faces. (This pattern does not apply to 2D smeared reinforcing element REINF263.)

Required input:

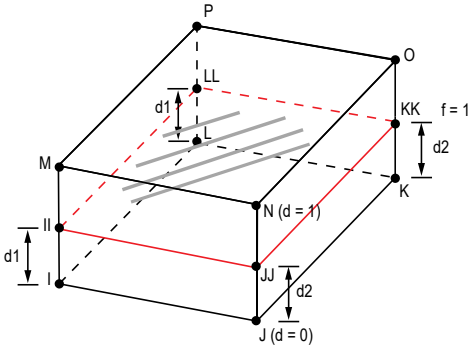
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

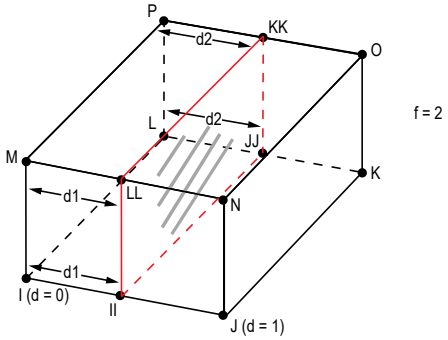
V1 (or *F*) -- The number to indicate the base element face. The default value is 1.

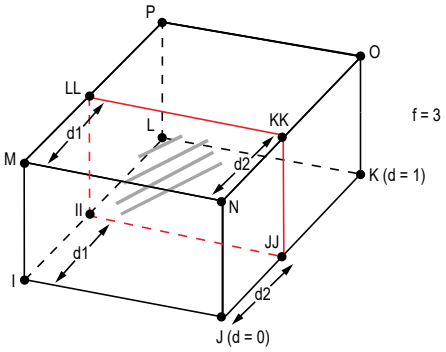
V2 (or *d1*) -- The normalized distance from the layer to the specified base element face. Valid values for *d1* are 0.0 through 1.0. The default value is 0.5.

V3 (or *d2*) -- The normalized distance from corners JJ and KK of the layer to the specified base element face (applicable to 8-node or 20-node solid elements only). Valid values for *d2* are 0.0 through 1.0. The default value is *d1*.

When applied to 8-node or 20-node solid elements:





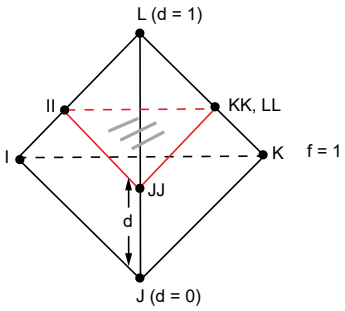


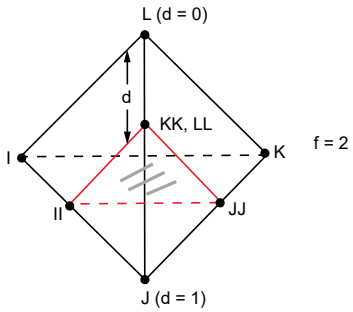
where:

d1 = distance II and LL

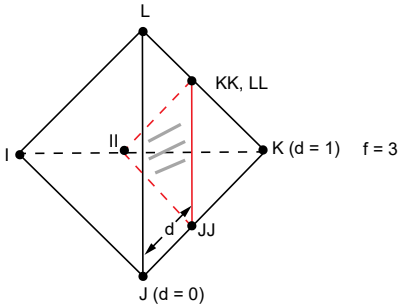
d2 = distance JJ and KK

When applied to tetrahedral elements:

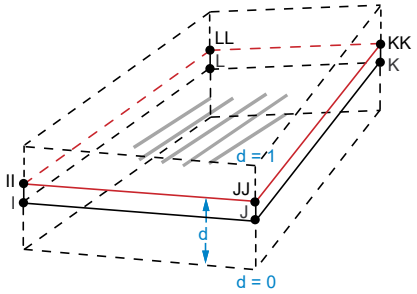




Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When applied to 3D shell elements:



Shells

Type: SHELL

Shell sections are referenced by the SHELL131, SHELL132, SHELL181, SOLID185 Layered Solid, SOLID186 Layered Solid, SOLSH190, SHELL208, SHELL209, SOLID278 Layered Solid, SOLID279 Layered Solid, and SHELL281 elements.

Data to provide in the value fields:

TK, *MAT*, *THETA*, *NUMPT*, `LayerName`

where:

TK = Thickness of shell layer. Zero thickness (not valid for SHELL131 and SHELL132) indicates a dropped layer. The sum of all layer thicknesses must be greater than zero. The total thickness can be tapered via the **SECFUNCTION (p. 1678)** command.

MAT = Material ID for layer (any current-technology material model is available for SHELL181, SOLID185 Layered Solid, SOLID186 Layered Solid, SOLSH190, SHELL208, SHELL209, SOLID278 Layered Solid and SOLID279 Layered Solid [including `UserMat`], and SHELL281). *MAT* is required for a composite (multi-layered) laminate. For a homogeneous (single-layered) shell, the default is the element material attribute. You can also address multiple reference temperatures (**TREF (p. 2013)** and/or **MP (p. 1160)**,REFT).

THETA = Angle (in degrees) of layer element coordinate system with respect to element coordinate system (ESYS).

NUMPT = Number of integration points in layer. The user interface offers 1, 3 (default), 5, 7, or 9 points; however, you can specify a higher number on the **SECDATA (p. 1655)** command. The integration rule used is Simpson's Rule. (*NUMPT* is not used by SHELL131 and SHELL132.)

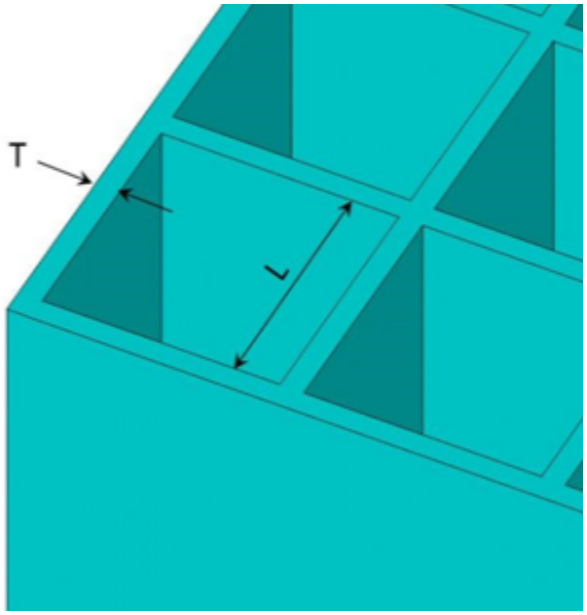
Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Supports

Type: SUPPORT

Support sections are referenced by SOLID185 and SOLID186 elements.

Type: SUPPORT, Subtype: BLOCK



Data to provide in the value fields for *Subtype* = BLOCK:

T, *L*

where:

T = Thickness of the block wall.

L = Spacing distance of the block walls.

Type: SUPPORT, Subtype: ASEC

Data to provide in the value fields for *Subtype* = ASEC:

Thermal	Structural
VAL1 = Multiplication factor for KXX	*VAL1* = Multiplication factor for EX
VAL2 = Multiplication factor for KYY	*VAL2* = Multiplication factor for EY
VAL3 = Multiplication factor for KZZ	*VAL3* = Multiplication factor for EZ
VAL4 = Multiplication factor for DENS	*VAL4* = Multiplication factor for GXY
	VAL5 = Multiplication factor for GYZ
	VAL6 = Multiplication factor for GXZ

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal	Structural
	VAL7 = Multiplication factor for ALPX *VAL8* = Multiplication factor for DENS

The multiplication factors are homogenization factors, and in each direction reflect the ratio of the support area projected onto the area of a fully solid support.

Values default to 1.0.

Y and Z values default to X values.

GXY value defaults to EX value, GYZ to EY, and GXZ to EZ.

Taper

Type: TAPER

Tapered sections are referenced by BEAM188, BEAM189 and ELBOW290 elements. After specifying the tapered section type (**SECTYPE (p. 1697)**,,TAPER), issue separate **SECDATA (p. 1655)** commands to define each end of the tapered beam or pipe.

Data to provide in the value fields:

Sec_IDn, *XLOC*, *YLOC*, *ZLOC*

where:

Sec_IDn = Previously defined beam or pipe section at ends 1 and 2.

XLOC, *YLOC*, *ZLOC* = The location of Sec_ID*n* in the global Cartesian coordinate system.

For more information about tapered beams and pipes, including assumptions and example command input, see Defining a Tapered Beam or Pipe in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Axis>Add

Main Menu>Preprocessor>Sections>Axis>Edit

Main Menu>Preprocessor>Sections>Beam>Common Sections

Main Menu>Preprocessor>Sections>Beam>Taper Sections>By Picked Nodes

Main Menu>Preprocessor>Sections>Beam>Taper Sections>By XYZ Location

Main Menu>Preprocessor>Sections>Contact>Add

Main Menu>Preprocessor>Sections>Contact>Edit

Main Menu>Preprocessor>Sections>Joints>Add / Edit

Main Menu>Preprocessor>Sections>Link>Add

Main Menu>Preprocessor>Sections>Link>Edit

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Sections>Pipe>Add

Main Menu>Preprocessor>Sections>Pipe>Edit

Main Menu>Preprocessor>Sections>Reinforcing>Add / Edit

Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit

SECFUNCTION, *TABLE*, *PATTERN*

Specifies shell section thickness as a tabular function.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TABLE

Name of table parameter or array parameter for specifying thickness.

PATTERN

Interpretation pattern for array parameters.

Notes

The **SECFUNCTION (p. 1678)** command is associated with the section most recently defined via the **SECTYPE (p. 1697)** command.

A table parameter can define tabular thickness as a function of coordinates. Alternatively, you can use an array parameter (indexed by node number) that expresses the function to be mapped. (See *PATTERN* of NOD2 for array content.) Enclose the table or array name in percent signs (%) (**SECFUNCTION (p. 1678)**,%`tablename`%).

Issue the ***DIM (p. 530)** command to define a table or array.

The table or array defines the total shell thickness at any point in space. In multilayered sections, the total thickness and each layer thickness are scaled accordingly.

The Function Tool is a convenient way to define your thickness tables.

Refer to the ***DIM (p. 530)** command for interpreting a table in a local coordinate system.

Specifying the *PATTERN* Value

When *PATTERN* = NODE, *TABLE* should be a 1D array parameter (indexed by node number) that expresses the function to be mapped.

When *PATTERN* = NOD2, *TABLE* should be a 2D array parameter (where column 1 contains node numbers and column 2 contains the corresponding thicknesses) that expresses the function to be mapped.

Specify *PATTERN* when *TABLE* is an *array parameter* only (and not when it is a table parameter or a single value).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit

SECJOINT, *DataType*, *Val1*, *Val2*, *Val3*, *Val4*, *Val5*, *Val6*

Defines local coordinate systems at joint element nodes and other data for joint elements.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DataType

The type of joint element data being defined:

LSYS

Define local coordinate systems at the nodes that form the MPC184 joint element (default).

RDOF

Define the relative degrees of freedom to be fixed for an MPC184-General joint element.

PITC

Define the pitch of an MPC184-Screw joint element.

FRIC

Define the geometric quantities required for Coulomb frictional behavior in the MPC184-Revolute, MPC184-Slot, MPC184-Translational, or MPC184-Spherical joint element.

PNLT

Define the penalty factors for the penalty-based joint element formulation.

SPWE

Define the spring-damper location for the MPC184-Spotweld joint element.

Val1, *Val2*, *Val3*, *Val4*, *Val5*, *Val6*

The values of *Val1* through *Val6* vary according to the specified *DataType*:

If *DataType* = LSYS (or blank), *Val1* and *Val2* are defined as follows:

Val1 = Local coordinate system at node I.

Val2 = Local coordinate system at node J.

(*Val3* through *Val6* are not used.)

Alternatively, a local coordinate system based on the vector from node I to node J of the joint element can be defined as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val1 = IJX (or IJY or IJZ), where:

IJX = vector from node I to node J is considered as the local X-axis at node I.

IJY = vector from node I to node J is considered as the local Y-axis at node I.

IJZ = vector from node I to node J is considered as the local Z-axis at node I.

(*Val2* through *Val6* are not used.)

A local coordinate system is constructed internally based on this vector from node I to J. The program selects the other two axes such that an orthonormal coordinate system is generated. This local coordinate system is used at both the nodes of the joint element in subsequent calculations.

If *DataType* = RDOF, *Val1* through *Val6* are the relative degrees of freedom to be fixed for a general joint element:

1 = UX

2 = UY

3 = UZ

4 = ROTX

5 = ROTY

6 = ROTZ

You can input the degrees of freedom in any order.

If *DataType* = PITC, *Val1* is the pitch of the screw joint element. The pitch is defined as the ratio of relative axial displacement (length units) to relative rotation (in radians). (*Val2* through *Val6* are not used.)

If *DataType* = FRIC, *Val1* through *Val3* are defined as follows (and *Val4* through *Val6* are not used):

For Revolute Joint:

Val1 = Outer radius

Val2 = Inner radius

Val3 = Effective length

For Translational Joint:

Val1 = Effective length

Val2 = Effective radius

For Spherical Joint:

Val1 = Effective radius

If *DataType* = PNLT, *Val1* and *Val2* are defined as follows (and *Val3* through *Val6* are not used):

Val1

Constraint type:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DISP - Displacement-based constraint

ROT - Rotation-based constraint

Val2

Scaling factor (positive number) or penalty factor (negative number) used with the penalty-based joint element formulation:

> 0 -- The number is used as a scaling factor to scale the internally calculated penalty values.

< 0 -- The absolute value of the number is used as the penalty factor in calculations.

If *DataType* = SPWE, *Val1*, *Val2*, *Val3*, and *Val4* are defined as follows:

Val1

Spring-damper location:

RATIO -- Specify only *Val2* (0 <= *Val2* <=1), indicating the spring-damper location between nodes I and J. (*Val3* through *Val6* are not used.)

COORD -- Specify *Val2*, *Val3*, and *Val4*. The values represent the global (X,Y,Z) coordinates, respectively, of the spring-damper location. (*Val5* and *Val6* are not used.)

If *Val1* is not specified, the default is RATIO with *Val2* = 0.5.

Notes

This command defines additional section data for MPC184 joint elements. The values that you input are interpreted based on the most recently issued **SECTYPE (p. 1697)** command. To overwrite current values, issue another **SECJOINT (p. 1679)** command with the same *DataType* value.

SECJOINT (p. 1679),LSYS,IJX (or IJY or IJZ) is typically used in spotweld applications. The joint element nodes should not be coincidental when this command is used. If the nodes are coincidental, the default global coordinate system applies. Specify axes to conform to the type of joint element used; otherwise, unexpected behavior may occur as the local coordinate system was not properly selected.

SECJOINT (p. 1679),PNLT applies to penalty-based joints only (KEYOPT(2) = 1 on most joint elements) and overwrites the default penalty factor. The default penalty factor (common to all constraints of the joint element) is of the order of $10*E_{avg}$, where E_{avg} is the average Young's modulus of the elements attached to the joint element or deduced from the connections to other elements. The penalty factor can affect the constraint satisfaction as well as overall solution convergence.

Menu Paths

Main Menu>Preprocessor>Sections>Joints>Add / Edit

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/SECLIB,*Option*,*Path*

Sets the default section library path for the SECREAD (p. 1693) command.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

READ

Sets the read path (default).

STATUS

Reports the current section library path setting to the `Jobname.LOG` file.

Path

Defines the directory path from which to read section library files.

Notes

When the **SECREAD (p. 1693)** command is issued without a directory path, the command searches for a section library in the following order:

- The user's home directory
- The current working directory
- The path specified by the **/SECLIB (p. 1682)** command

Menu Paths

Main Menu>Preprocessor>Sections>Section Library>Library Path

SECLOCK,*dof*,*MINVALUE*,*MAXVALUE*,*dof*,*MINVALUE*,*MAXVALUE*,*dof*,*MINVALUE*,*MAXVALUE*

Specifies locks on the components of relative motion in a joint element.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

dof

Local degree of freedom to be locked.

MINVALUE

Low end of the range of allowed movement for the specified DOF.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MAXVALUE

High end of the range of allowed movement for the specified DOF.

Notes

Specify up to three degrees of freedom to be locked. Locks are activated when the limit values are reached, and further motion in that degree of freedom is frozen. If necessary, the command can be issued multiple times.

SECLOCK (p. 1682) is not available for spotweld joints.

Menu Paths

Main Menu>Preprocessor>Sections>Joints>Add / Edit

SECMODIF, *SECID*, *Keyword*

Modifies a pretension or joint section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

If *Keyword* = NORM, command format is:

SECMODIF (p. 1683),*SECID*, NORM, *NX*, *NY*, *NZ*, *KCN*

SECID

Unique section number. This number must already be assigned to a section.

NORM

Modify the pretension section normal direction.

NX, NY, NZ

The individual normal components to modify.

KCN

Coordinate system number: 0 = global Cartesian, 1 = global cylindrical), 2 = global spherical, 4 = working plane), 5 = global Y axis cylindrical, or *n* = an arbitrary reference number assigned to a coordinate system.

If *Keyword* = NAME, the command format is:

SECMODIF (p. 1683),*SECID*, NAME, *Name*

SECID

Unique section number. This number must already be assigned to a section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NAME

Change the name of the specified pretension section.

Name

The new name to be assigned to the pretension section.

If *Keyword* = JOIN, the command format is:

SECMODIF (p. 1683),*SECID*, JOIN, *Option*, *dispPnltVal*, *RotPnltVal*

SECID

Unique section number (which must already be assigned to a section), or unspecified. If no section number is specified, the penalty factors are applied to all joint sections associated with penalty-based joint elements only.

JOIN

Set command actions to apply to joint sections only.

Option

PNLT -- Modify penalty factors for the specified section.

dispPnltVal

Penalty value for displacement-based constraints:

> 0 -- The number is used as a scaling factor to scale the internally calculated penalty values.

< 0 -- The absolute value of the number is used as the penalty factor in calculations.

RotPnltVal

Penalty value for rotation-based constraints.

> 0 -- The number is used as a scaling factor to scale the internally calculated penalty values.

< 0 -- The absolute value of the number is used as the penalty factor in calculations.

Notes

SECMODIF (p. 1683) either modifies the normal for a specified pretension section, or changes the name of the specified pretension surface.

For joint sections associated with *penalty-based joint elements*, **SECMODIF (p. 1683)** modifies the penalty factors of a previously defined joint section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Sections>Pretension>Modify Name

Main Menu>Preprocessor>Sections>Pretension>Modify Normal

SECNUM, *SECID*

Sets the element section attribute pointer.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SECID

Defines the section ID number to be assigned to the subsequently-defined elements. Defaults to 1. See **SECTYPE (p. 1697)** for more information about the section ID number.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

SECOFFSET, *Location*, *OFFSET1*, *OFFSET2*, *CG-Y*, *CG-Z*, *SH-Y*, *SH-Z*

Defines the section offset for cross sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Location*, *OFFSET1*, *OFFSET2*, *CG-Y*, *CG-Z*, *SH-Y*, *SH-Z

The location of the nodes in the section. All are dependent on the type. See the "Notes" (p. 1685) section for information about the values for the various section types.

Notes

The **SECOFFSET (p. 1685)** command is divided into four types: Beams (p. 1686), Pipes (p. 1687), Shells (p. 1687), and Preintegrated General Shells (p. 1688).

The offsets defined by the **SECOFFSET (p. 1685)** command are associated with the section most recently defined using the **SECTYPE (p. 1697)** command. Not all **SECOFFSET (p. 1685)** location values are valid for each subtype.

For the thermal shell elements, SHELL131 and SHELL132, the node offset specified by **SECOFFSET (p. 1685)** is used in thermal contact analyses. Otherwise, the **SECOFFSET (p. 1685)** command has no effect on the solution for these elements and is used only for visualization purposes.

This command is not valid with thermal solid elements SOLID278 and SOLID279.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For beam elements BEAM188/BEAM189 and pipe elements PIPE288/PIPE289 in the XY plane, an offset is not allowed if it causes the elements to be nonsymmetric about the XY plane.

Beams

Type: BEAM

Argument data to provide:

Location, *OFFSETY*, *OFFSETZ*, *CG-Y*, *CG-Z*, *SH-Y*, *SH-Z*

Location

CENT --

Beam node will be offset to centroid (default).

SHRC --

Beam node will be offset to shear center.

ORIGIN --

Beam node will be offset to origin of the cross section.

USER --

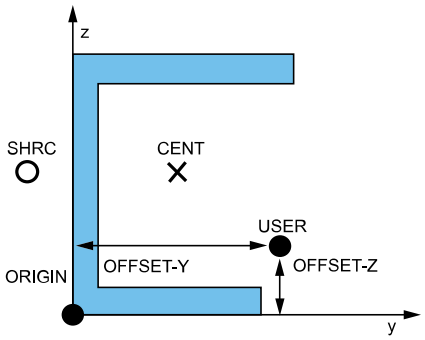
Beam node will be offset to the location specified by *OFFSETY* and *OFFSETZ*.

OFFSETY, OFFSETZ

Values that locate the node with respect to the default origin of the cross section when the *Location* argument is set to USER. Valid only when USER is set.

The following figure illustrates the offsets for a channel cross section, and shows the relative locations of SHRC and CENT.

Figure 15: Offsets for a CHAN Section Subtype



Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CG-Y*, *CG-Z*, *SH-Y*, *SH-Z

Override the program-calculated centroid and shear centroid locations.

Note:

This option should only be used by *advanced users* modeling composite cross sections.

Pipes

Type: PIPE

Argument data to provide:

OFFSETY, *OFFSETZ*

OFFSETY*, *OFFSETZ

Values that locate the node with respect to the center of the pipe.

Shells

Type: SHELL

Argument data to provide:

Location, *OFFSET*

Location

TOP --

Shell node will be offset to top of the section.

MID --

Shell node will be offset to midplane of the section (default).

BOT --

Shell node will be offset to bottom of the section.

USER --

Shell node will be offset to the location specified by *OFFSET*.

OFFSET

Value that locates the node with respect to the default origin (midplane) of the section. Valid only when *Location* = USER.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The offset alters only the *reference surface* of the shell elements (that is, where the nodes are located). It does not change the physical dimensions of the shell itself; the volume and mass remain constant when an offset is specified.

Preintegrated General Shells

Type: GENS

Argument data to provide:

Location, *OFFSET*

Location

MID --

Shell node will be offset to midplane of the section (default).

USER --

Shell node will be offset to the location specified by *OFFSET*.

OFFSET

Value that locates the node with respect to the default origin (midplane) of the section. Valid only when *Location* = USER.

The offset alters only the *reference surface* of the shell elements (that is, where the nodes are located).

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Common Sections

Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh

Main Menu>Preprocessor>Sections>Pipe>Add

Main Menu>Preprocessor>Sections>Pipe>Edit

Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit

SECPLOT, *SECID*, *VAL1*, *VAL2*, *VAL3*

Plots the geometry of a beam, pipe, elbow, shell, or reinforcing section to scale.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SECID

The section ID number (as defined via the **SECTYPE (p. 1697)** command).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL1, VAL2, VAL3

Values that control the information to be plotted. See the "Notes" (p. 1689) section of this command description for more information. For clarity, the labels *VAL1*, *VAL2*, and *VAL3* are renamed according to the section type.

Notes

SECPLOT (p. 1688) is valid for "Beams and Pipes" (p. 1689), "Shells" (p. 1691), "Reinforcings" (p. 1692), and ELBOW290.

SECPLOT (p. 1688) cannot display the plot of an ASEC (arbitrary section) subtype.

Beams and Pipes

Plots the geometry of the beam or pipe section to scale depicting the centroid, shear center, and origin. **SECPLOT (p. 1688)** also lists various section properties such as *Iyy*, *Iyz*, and *Izz*.

Data to be supplied in the value fields:

MESHKEY

Beam or pipe section mesh-display options:

0 --

Display section outline only.

1 --

Display beam or pipe section mesh.

For ELBOW290, specify *ELBKEY* to display the section mesh.

2 --

Display the section mesh with cell node numbers.

For ELBOW290, display a pipe section mesh with integration station numbers when *ELBKEY* is specified. (If internal fluid and insulation are not defined, this option works the same as specifying *MESHKEY* = 6 for ELBOW290.)

3 --

Display the section mesh with cell numbers.

For ELBOW290, display a pipe section mesh with layer numbers when *ELBKEY* is specified.

4 --

Display the section mesh with material numbers and colors.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 --

Display the section mesh with material colors only.

6 --

Display the section mesh with the RST node numbers or RST location numbers. RST nodes are section corner nodes where results are available. This option applies when the averaged results format (KEYOPT(15) = 0 for BEAM188, BEAM189, PIPE288, and PIPE289) is used.

For ELBOW290, RST location numbers are selected section-integration station numbers (where results are available). The numbers may vary based on *ELBKEY*.

7 --

Display the section mesh with the RST cell numbers or RST layer numbers. RST cells are section cells where results are available. This is applicable when the non-averaged results format (KEYOPT(15) = 1 for BEAM188, BEAM189, PIPE288, and PIPE289) is used.

For ELBOW290, RST layer numbers are section layers (where results are available).

Options 2 through 7 do not depict centroid and shear center, nor do they list section properties.

ELBKEY

Additional pipe-section mesh-display options for ELBOW290:

(Blank) --

Display as a standard pipe section.

0 --

Display the section mesh for the option to save the results at the bottom of the bottom layer and the top of the top layer.

1 --

Display the section mesh for the option to save the results at the top and bottom surfaces of each layer.

2 --

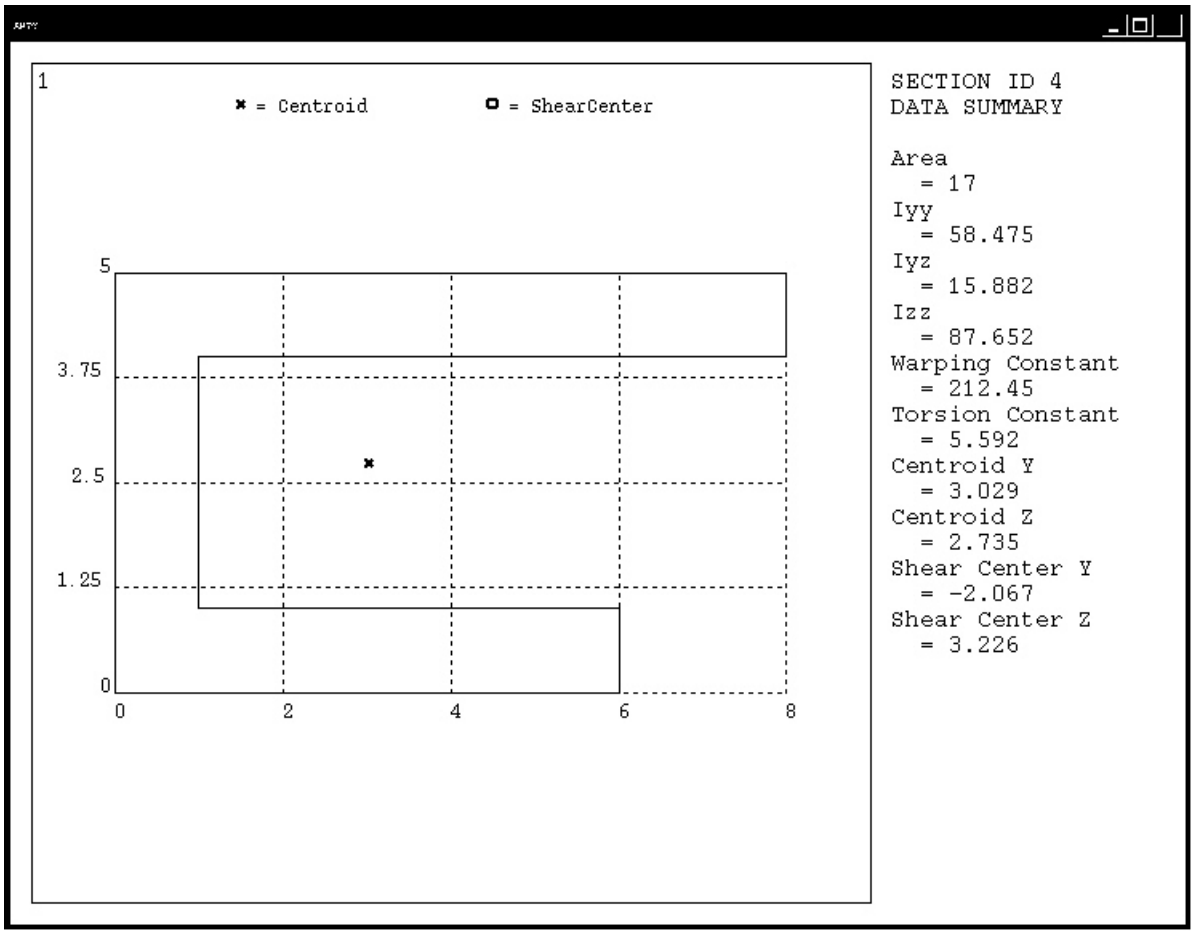
Display the section mesh for the option to save the results at the top, bottom, and mid-side surfaces of each layer.

When *ELBKEY* is specified, the display for *MESHKEY* = 2 or 6 changes with respect to the *ELBKEY* value. The display for MESHKEY = 0, 1, 3, 4, 5, or 7 is the same regardless of the *ELBKEY* value.

The specified *ELBKEY* must be appropriate for the selected ELBOW290 layer-data storage option (KEYOPT(8)).

Following is a sample section plot for the beam section type:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Shells

Plots the layer arrangement of the shell section showing the layer material and orientation.

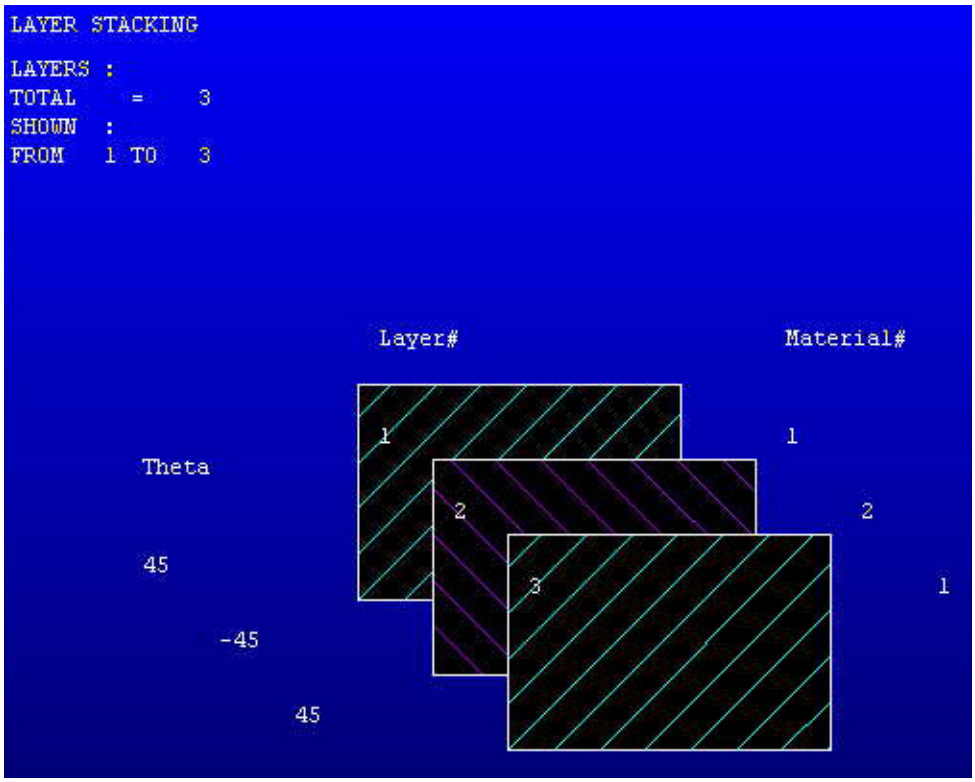
Data to be supplied in the value fields:

LAYR1, LAYR2

The range of layer numbers to be displayed. If *LAYR1* is greater than *LAYR2*, a reversed order display is produced. Up to 20 layers may be displayed at the same time. *LAYR1* defaults to 1. *LAYR2* defaults to *LAYR1* if *LAYR1* is input or to the number of layers (or to 19+*LAYR1*, if smaller) if *LAYR1* is not input.

Following is a sample section plot for the shell section type:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Reinforcings

Plots the arrangement of a reinforcing (p. 1665) section within the base element.

Data to be supplied in the value fields:

REINF1, REINF2, OVERLAY

REINF1, REINF2 -- The numerical range of reinforcings to be displayed. The default *REINF1* value is 1. The default *REINF2* value is the number of reinforcings.

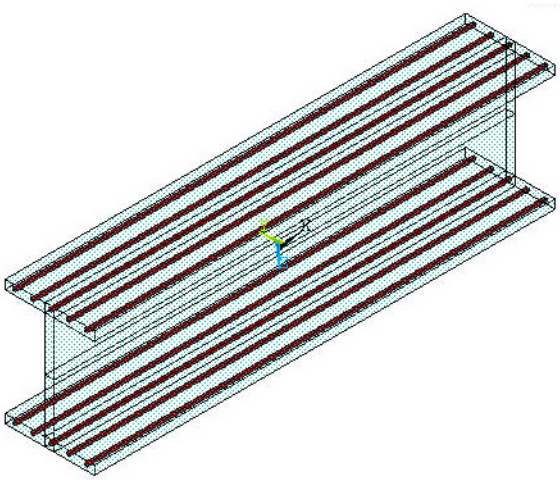
OVERLAY -- The section ID of the base element within which to display the reinforcing section. The section appears translucent and the reinforcing section is solid. Valid values are:

- SOLID -- Display a translucent solid block over the reinforcing section
- *SECID* -- A number corresponding to a specific section ID of the base element.

If no *OVERLAY* value is specified, the program displays the reinforcing section only.

Following is a sample section plot for the reinforcing section type:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For more information about reinforcing, see the documentation for the **SECDATA (p. 1655)** command, and the REINF264 and REINF265 elements.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Plot Section

Main Menu>Preprocessor>Sections>Pipe>Plot Section

Main Menu>Preprocessor>Sections>Reinforcing>Plot Section

Main Menu>Preprocessor>Sections>Shell>Lay-up>Plot Section

SECREAD,*Fname*,*Ext*,--,*Option*

Reads a custom section library or a user-defined section mesh into Mechanical APDL.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

Section library file name and directory path containing the section library file (248 characters maximum, including directory). If you do not specify a directory path, it will default to your working directory and you can use all 248 characters for the file name.

When the **SECREAD (p. 1693)** command is given without a directory path, the command searches for a section library in the following order:

- The user's home directory
- The current working directory
- The path specified by the **/SECLIB (p. 1682)** command

The file name defaults to Jobname if *Fname* is left blank.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ext

Filename extension (eight-character maximum).

The extension defaults to SECT if *Ext* is left blank.

--

Unused field.

Option

LIBRARY

Reads in a library of sections and their associated section data values; the default. A section library may be created by editing the section-defining portions of the `Jobname.log` file and saving it with a `.sect` suffix.

MESH

Reads in a user mesh section file containing the cell connectivity, cell flags, and nodal coordinates for the current beam section of subtype MESH as defined by **SECTYPE (p. 1697)**. See the "Notes" (p. 1694) section of this command description for details about user mesh section files. **SECWRITE (p. 1703)** builds mesh files based on 2D models you create.

Notes

The **SECREAD (p. 1693)** command operates on the section specified via the most recently issued **SECTYPE** command. Issue a separate **SECREAD (p. 1693)** command for each section ID that you want to read in.

Sample User Section Cell Mesh File

Here are excerpts from a sample user section mesh file for a section with 75 nodes, 13 cells, and 9 nodes per cell for a two-hole box section. Illustrations of the two-hole box section and the cell mesh for it appear later in this command description.

First Line:

```
75   13
```

Cells Section:

```
      1   3   25   23   2   14   24   12   13   1
      3   5   27   25   4   16   26   14   15   1
      5   7   29   27   6   18   28   16   17   1
      7   9   31   29   8   20   30   18   19   1
      9   11  33   31   10  22   32   20   21   1
      23  25  45   43   24  36   44   34   35   1
...
      49  51  73   71   50  62   72   60   61   1
      51  53  75   73   52  64   74   62   63   1
0     0.0          0.0
0     0.025        0.0
0     0.05         0.0
0     5.0175       0.0
```

Nodes Section:

```
...
0    19.98        10.00
0    20.00        10.00
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The mesh file is divided into three sections: the First Line, the Cells Section, and the Nodes Section. Here are brief descriptions of the contents of each.

First Line: The First Line defines the number of nodes and the number of cells for the mesh.

Cells Section: The Cells Section contains as many lines as there are cells. In this example, there are thirteen cells, so there are thirteen lines in this section. In each line, the number "1" that follows the cell connectivity information is the material number.

Cell nodal connectivity must be given in a counterclockwise direction, with the center node being the ninth node. For details, see Figure 17: Cell Mesh for the Two-hole Box Section (p. 1695).

Nodes Section: The Nodes Section contains as many lines as there are nodes. In this example, there are 75 nodes, so there are a total of 75 lines in this section. Each node line contains the node's boundary flag, the Y coordinate of the node, and the Z coordinate of the node. Currently, all node boundary flags appear as 0s in a cell mesh file (as illustrated in Figure 16: Two-hole Box Section (p. 1695)). Since all node boundary flags are 0, **SECREAD (p. 1693)** ignores them when it reads a cell mesh file into Mechanical APDL.

There cannot be any gaps in the node numbering of a cell mesh. The nodes in a cell mesh must be numbered consecutively, with the first node having a node number of 1, and the last node having a node number that is equal to the maximum number of nodes in the cell mesh.

Figure 16: Two-hole Box Section

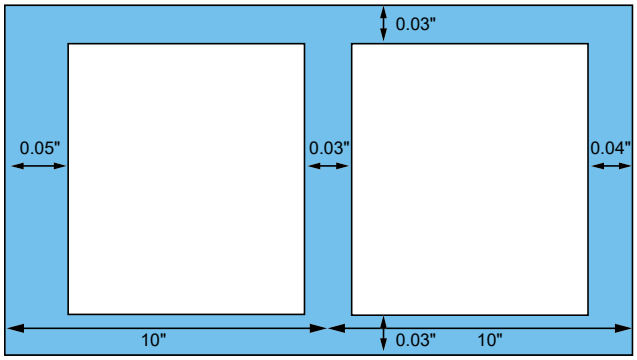
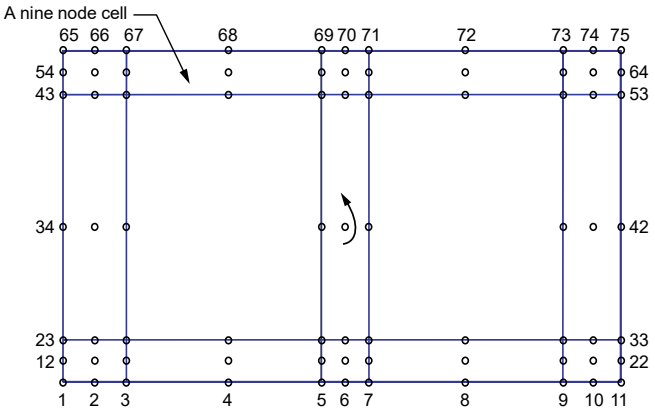


Figure 17: Cell Mesh for the Two-hole Box Section



Menu Paths

Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh

Main Menu>Preprocessor>Sections>Section Library>Import Library

---

SECSTOP,*dof*,*MINVALUE*,*MAXVALUE*,*dof*,*MINVALUE*,*MAXVALUE*,*dof*,*MINVALUE*,*MAXVALUE*

Specifies stops on the components of relative motion in a joint element.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

dof

Local degree of freedom to be stopped.

MINVALUE

Low end of the range of allowed movement for the specified degree of freedom.

MAXVALUE

High end of the range of allowed movement for the specified degree of freedom.

Notes

Restricts motion in a degree of freedom. Motion beyond the *MINVALUE* or *MAXVALUE* is prevented. (Motion away from a limit is allowed.) You can specify up to three stops. If necessary, you can issue the command multiple times.

SECSTOP (p. 1696) is not available for spotweld joints.

Menu Paths

Main Menu>Preprocessor>Sections>Joints>Add / Edit

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SECTYPE, *SECID*, *Type*, *Subtype*, *Name*, *REFINEKEY*

Associates section type information with a section ID number.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1702)

SECID

Section identification number. If *SECID* is blank or zero, the *SECID* number is incremented by one from the highest section ID number currently defined in the database. (See Notes (p. 1701) for *SECID* input specific to general contact.)

Type

AXIS

Define the axis for a general axisymmetric section.

BEAM

Defines a beam section. This option has a *Subtype*.

COMB

Defines a composite (temperature-dependent) beam section. This option has a *Subtype*.

CONTACT

Defines a contact section. This option has a *Subtype*.

GENB

Defines a nonlinear general (temperature-dependent) beam section. This option has a *Subtype*.

GENS

Defines a preintegrated general (temperature-dependent) shell section.

JOINT

Defines a joint section. This option has a *Subtype*.

LINK

Defines a link section.

PIPE

Defines a pipe section.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PRETENSION

Defines a pretension section.

REINF

Defines a reinforcing section. This option has a *Subtype*.

SHELL

Defines a shell section.

SUPPORT

Additive manufacturing support. This option has a *Subtype*.

TAPER

Defines a tapered beam or pipe section. The sections at the end points must be topologically identical.

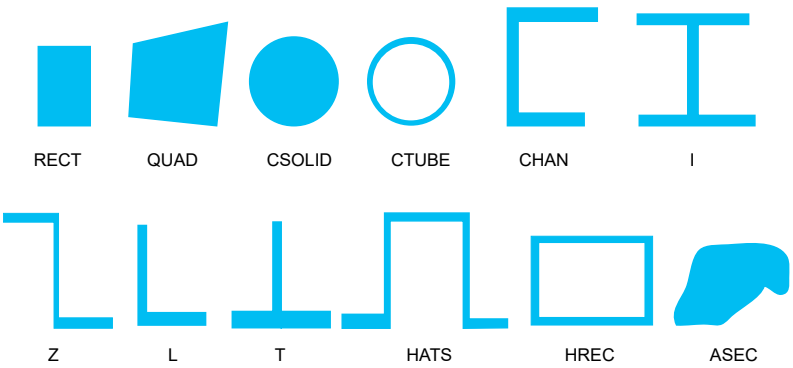
Subtype

When *Type* = BEAM, the possible beam sections that can be defined for *Subtype* are:

RECT	Rectangle
QUAD	Quadrilateral
CSOLID	Circular solid
CTUBE	Circular tube
CHAN	Channel
I	I-shaped section
Z	Z-shaped section
L	L-shaped section
T	T-shaped section
HATS	Hat-shaped section
HREC	Hollow rectangle or box
ASEC	Arbitrary section -- integrated cross-section inertia properties supplied by user
MESH	User-defined mesh -- see the **SECREAD (p. 1693)** command for more information about this data

The following figure shows the shape of each cross section subtype:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When *Type* = COMB, the only possible composite-beam section that can be defined for *Subtype* is:

MATRIX	Matrix.

When *Type* = CONTACT, the possible contact sections that can be defined for *Subtype* are:

CIRCLE	Geometry correction for a portion of a circle (or nearly a circle).
SPHERE	Geometry correction for a portion of a spherical (or nearly spherical) surface.
CYLINDER	Geometry correction for a portion of a revolute (or nearly revolute) surface.
NORMAL	Geometry correction to specify a user-defined contact surface normal.
BOLT	Geometry correction for a bolt thread surface.
RADIUS	Equivalent beam/edge radius for 3D beam-to-beam or 3D edge-to-edge contact in a general contact definition; or radii associated with rigid target segments in a general contact definition.

When *Type* = GENB, the possible nonlinear general beam sections that can be defined for *Subtype* are:

ELASTIC	The generalized-stress/generalized-strain relationship is elastic (linear or nonlinear).
PLASTIC	The generalized-stress/generalized-strain relationship is elastoplastic (and allows for permanent deformation).

When *Type* = JOINT, the possible joint sections that can be defined for *Subtype* are:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UNIV	Universal joint
REVO	Revolute joint
SLOT	3D Slot joint
PINP	Point-in-plane joint
PRIS	Translational joint
CYLI	Cylindrical joint
PLAN	Planar joint
WELD	Weld joint
ORIE	Orient joint
SPHE	Spherical joint
GENE	General joint
SCRE	Screw joint

When *Type* = REINF, the possible reinforcing sections that can be defined for *Subtype* are:

DISC	Discrete reinforcing. The reinforcing fibers are arbitrarily oriented and modeled individually.
SMEAR	Smeared reinforcing. The reinforcing fibers are homogeneous and defined as a membrane.
GCON	General connection reinforcing. This subtype is generated by **EEMBED (p. 595)** for REINF265 elements when using those elements to model the interior interfaces between embedded elements and base elements. You can expose this subtype (**SLIST (p. 1795)**) but cannot manually specify or modify it.

When *Type* = SUPPORT, the possible support sections that can be defined for *Subtype* are:

BLOCK	Block support structure.
ASEC	User-provided factors.

Name

An eight-character name for the section. *Name* can be a string such as "W36X210" or "HP13X73" for beam sections. Section name can consist of letters and numbers, but cannot contain punctuation, special characters, or spaces.

REFINEKEY

Sets mesh refinement level for thin-walled beam sections. Valid values are 0 (the default - no mesh refinement) through 5 (high level of mesh refinement). This value has meaning only when *Type* = BEAM.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

SECTYPE (p. 1697) sets the section ID number, section type, and subtype for a section. A previously-defined section with the same identification number will be redefined. The geometry data describing this section type is defined by a subsequent **SECDATA (p. 1655)** command. Define the offsets (if applicable) by a subsequent **SECOFFSET (p. 1685)** command. The **SLIST (p. 1795)** command lists the section properties, and the **SECPLOT (p. 1688)** command displays the section to scale. The **SECNUM (p. 1685)** command assigns the section ID number to any subsequently-defined elements.

When defining a section for contact elements (*Type* = CONTACT) that are used in a general contact definition, a section number representing a general contact surface can be specified. Alternatively, you may define a subset of a region by inputting a valid label for *SECID* (ALL_EDGE, ALL_FACE, ALL_VERT, ALL_TOP,or ALL_BOT), or by inputting a node component name with or without a component name extension (_EDGE, _FACE, _VERT, _TOP, or _BOT). For more information, see Applying Surface Geometry Correction in the *Contact Technology Guide*.

Beam Section Considerations

For a beam section (*Type* = BEAM), a subsequent **SECDATA (p. 1655)** command builds a numeric model using a nine-node cell for determining the properties (*Ixx*, *Iyy*, etc.) of the section and for the solution to the Poisson's equation for torsional behavior. See Beam Analysis and Cross Sections in the *Structural Analysis Guide* for examples using the section commands.

For a nonlinear general beam section (*Type* = GENB), the *Subtype* and *REFINEKEY* options do not apply. Subsequent commands are necessary to define the section: **BSAX (p. 271)**, **BSM1 (p. 273)**, **BSM2 (p. 274)**, **BSTQ (p. 278)**, **BSS1 (p. 276)**, **BSS2 (p. 277)**, **BSMD (p. 272)**, and **BSTE (p. 278)** are available. All other section commands are ignored for this section type.

For a preintegrated composite-beam section (*Type* = COMB), the *REFINEKEY* options do not apply. Subsequent commands are necessary to define the section: **CBTMP (p. 290)**, **CBMX (p. 288)**, **CBMD (p. 287)**, and **CBTE (p. 289)** are available. All other section commands are ignored for this section type.

Tapered Section Considerations

For a tapered beam or pipe section (*Type* = TAPER), two subsequent **SECDATA (p. 1655)** commands are required (one for each end section). Section ends must be topologically identical (same *Subtype*, number of cells and material IDs). For a tapered pipe section, end sections must have the same number of cells around the circumference and along the pipe wall, and the same shell section ID for a composite pipe wall.

Shell Section Considerations

For a preintegrated general shell section (*Type* = GENS), the *Subtype* and *REFINEKEY* options do not apply. Subsequent commands are necessary to define the section: **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, **SSBT (p. 1846)**, and **SSPM (p. 1855)** are available. All other section commands are ignored for this section type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pretension Section Considerations

The PRETENSION section options of the **SECTYPE (p. 1697)** and **SECDATA (p. 1655)** commands are documented primarily to aid your understanding of the data written by the **CDWRITE (p. 293)** command. Ansys, Inc. recommends that you generate pretension sections via the **PSMESH (p. 1543)** command.

Reinforcing Section Considerations

For a reinforcing section (*Type* = REINF), each subsequent **SECDATA (p. 1655)** command defines the material, geometry, and orientation of one discrete reinforcing fiber (*Subtype* = DISC) or one smeared reinforcing surface (*Subtype* = SMEAR). When referenced by a MESH200 element, only one **SECDATA (p. 1655)** command is valid.

Support Section Considerations

A subsequent **SECDATA (p. 1655)** command defines the geometry data describing this section type.

Displaying Elements with Section Definition

To display elements with shapes determined from the section definition, issue the **/ESHAPE (p. 665)** command.

Product Restrictions

Ansys Mechanical Pro

SECTYPE (p. 1697),COMB is not valid.

Menu Paths

Main Menu>Preprocessor>Sections>Axis>Add

Main Menu>Preprocessor>Sections>Axis>Edit

Main Menu>Preprocessor>Sections>Beam>Common Sections

Main Menu>Preprocessor>Sections>Beam>Composite Sections

Main Menu>Preprocessor>Sections>Beam>Custom Sections>Read Sect Mesh

Main Menu>Preprocessor>Sections>Beam>NL Generalized

Main Menu>Preprocessor>Sections>Beam>Taper Sections>By Picked Nodes

Main Menu>Preprocessor>Sections>Beam>Taper Sections>By XYZ Location

Main Menu>Preprocessor>Sections>Contact>Add

Main Menu>Preprocessor>Sections>Contact>Edit

Main Menu>Preprocessor>Sections>Joints>Add / Edit

Main Menu>Preprocessor>Sections>Link>Add

Main Menu>Preprocessor>Sections>Link>Edit

Main Menu>Preprocessor>Sections>Pipe>Add

Main Menu>Preprocessor>Sections>Pipe>Edit

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Sections>Reinforcing>Add / Edit

Main Menu>Preprocessor>Sections>Shell>Lay-up>Add / Edit

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

SECWRITE, *Fname*, *Ext*, --, *ELEM_TYPE*

Creates an ASCII file containing user mesh section information.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name defaults to Jobname if *Fname* is left blank.

Ext

Filename extension (eight-character maximum).

The extension defaults to SECT if *Ext* is left blank.

--

Unused field.

ELEM_TYPE

Element type attribute pointer (**ET (p. 686)**) for the elements that are part of the section. See **SECREAD (p. 1693)** for a detailed description.

Notes

Before creating a user mesh file, first create a model using 2D meshing. Use PLANE183 or MESH200 with KEYOPT(1) = 7 (quadrilateral with 8 nodes option) to model the cells. **SECWRITE (p. 1703)** creates an ASCII file that contains information about the nodes and cells that describe a beam section. For detailed information on how to create a user mesh file, see Creating Custom Cross Sections with a User-defined Mesh in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Sections>Beam>Custom Sections>Write From Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SED, *SEDX*, *SEDY*, *SEDZ*, *Cname*

Defines the excitation direction for response spectrum and PSD analyses.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SEDX*, *SEDY*, *SEDZ

Global Cartesian coordinates of a point that defines a line (through the origin) corresponding to the excitation direction. For example: 0.0, 1.0, 0.0 defines global Y as the spectrum direction.

Cname

The component name corresponding to the group of excited nodes. Only applies to base excitation multi-point response spectrum analysis (**SPOPT (p. 1837)**, MPRS) and power spectral density analysis (**SPOPT (p. 1837)**, PSD). Defaults to no component.

Notes

In single-point response spectrum analysis (**SPOPT (p. 1837)**,SPRS), the excitation direction without rocking (**ROCK (p. 1614)**) is normalized to one so that the *SEDX*, *SEDY*, and *SEDZ* values do not scale the spectrum. The excitation direction with rocking is not normalized. The *SEDX*, *SEDY*, and *SEDZ* values must be consistent with the linear components of *OMX*, *OMY*, and *OMZ* values on the **ROCK (p. 1614)** command. The calculated direction then scales the spectrum. For more information, see Participation Factors and Mode Coefficients.

In multi-point response spectrum analysis (**SPOPT (p. 1837)**,MPRS) and power spectral density analysis (**SPOPT (p. 1837)**,PSD), the excitation direction is normalized to one so that the *SEDX*, *SEDY*, and *SEDZ* values do not scale the spectrum. The component name (*Cname*) is required. The constraints corresponding to the excitation direction are applied to the component nodes.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Settings

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>PSD>Settings

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings

Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Settings

Main Menu>Solution>Load Step Opts>Spectrum>PSD>Settings

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SEDLIST, *Sename*, *KOPT*

Lists the DOF solution of a superelement after the use pass.

PREP7 (p. 22): Superelements (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sename

Name of the superelement in `Jobname.dsub` to be listed. If a number, it is the element number of the superelement as used in the use pass. If ALL, list results for all superelements.

KOPT

List key:

0

List summary data only.

1

List full contents. Be aware that the listing may be extensive.

Notes

Lists the degree of freedom solution of a superelement after the substructure use pass. Results may be listed for any superelement on `File.dsub`.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>List Results>Superelem DOF

Utility Menu>List>Results>Superelem DOF Solu

SEEXP, *Sename*, *Usefil*, *Imagky*, *Expopt*

Specifies options for the substructure expansion pass.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sename

The name (case-sensitive) of the superelement matrix file created by the substructure generation pass (*Sename* .SUB). Defaults to the initial jobname `File`. If a number, it is the element number of the superelement as used in the use pass.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Usefil

The name of the file containing the superelement degree-of-freedom (DOF) solution created by the substructure use pass (*Usefil*.DSUB).

Imagky

Key to specify use of the imaginary component of the DOF solution. Applicable only if the use pass is a harmonic (**ANTYPE (p. 162)**,HARMIC) analysis:

OFF

Use real component of DOF solution (default).

ON

Use imaginary component of DOF solution.

Note:

If all solutions are to be expanded (**NUMEXP (p. 1293)**,ALL), *Imagky* is ignored and both the real and imaginary solutions are expanded.

Expopt

Key to specify whether the superelement (**ANTYPE (p. 162)**,SUBSTR) expansion pass (**EXPASS (p. 714)**,ON) should transform the geometry:

OFF

Do not transform node or element locations (default).

ON

Transform node or element locations in the FE geometry record of the .rst results file.

Notes

Specifies options for the expansion pass of the substructure analysis (**ANTYPE (p. 162)**,SUBSTR). If used in SOLUTION, this command is valid only within the first load step.

If you specify geometry transformation (*Expopt* = ON), you must retrieve the transformation matrix (if it exists) from the specified .SUB file. The command updates the nodal X, Y, and Z coordinates to represent the transformed node locations. The *Expopt* option is useful when you want to expand superelements created from other superelements (via **SETRAN (p. 1730)** or **SESYMM (p. 1718)** commands). For more information, see Superelement Expansion in Transformed Locations and Plotting or Printing Mode Shapes.

This command is also valid in **/PREP7 (p. 1479)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>ExpansionPass>Single Expand>Expand Superelem

Main Menu>Solution>Load Step Opts>ExpansionPass>Single Expand>Expand Superelem

---

/SEG, *Label*, *Aviname*, *DELAY*

Allows graphics data to be stored in the local terminal memory.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Storage key:

SINGL

Store subsequent display in a single segment (overwrites last storage).

MULTI

Store subsequent displays in unique segments (**ANIM (p. 142)**).

DELET

Delete all currently stored segments.

OFF

Stop storing display data in segments.

STAT

Display segment status.

PC

This option only applies to PC versions of Mechanical APDL and only when animating via the AVI movie player (**/DEVICE (p. 523)**,ANIM,2). The command appends frames to the `File.avi`, so that the animation goes in both directions (that is, forward--backward--forward). You must have a current animation file to use this option.

Aviname

Name of the animation file that will be created when each frame is saved. The `.avi` extension is applied automatically. Defaults to `Jobname.avi` if no filename is specified.

DELAY

Delay factor between each frame, in seconds. Defaults to 0.015 seconds if no value is specified.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

No segment storage.

Notes

Allows graphics data to be stored in the terminal local memory (device-dependent). Storage occurs concurrently with the display.

Although the information from your graphics window is stored as an individual segment, you cannot plot directly (**GPLOT (p. 866)**) from the segment memory.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Redirect Plots>Delete Segments

Utility Menu>PlotCtrls>Redirect Plots>Segment Status

Utility Menu>PlotCtrls>Redirect Plots>To Segment Memory

SELIST, *Sename*, *KOPT*, *KINT*

Lists the contents of a superelement matrix file.

PREP7 (p. 22): Superelements (p. 34)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sename

The name (case-sensitive) of the superelement matrix file created by the substructure generation pass (*Sename* . SUB). Defaults to the current Jobname. If a number, it is the element number of the superelement as used in the use pass.

KOPT

List key:

0

List summary data only.

1

List contents, except load vectors and matrices.

2

List contents, except matrices.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3

List full contents. Be aware that the listing may be extensive.

KINT

Integer printout format key:

OFF

Default.

ON

Long format for large integers.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Superelem Data

SELM

Specifies "Superelements" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Superelements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SELTOL, *Toler*

Sets the tolerance for subsequent select operations.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Toler

Tolerance value. If blank, restores the default tolerance logic.

Notes

For selects based on non-integer numbers (e.g. coordinates, results, etc.), items within the range VMIN - *Toler* and VMAX + *Toler* are selected, where VMIN and VMAX are the range values input on the xSEL commands (**ASEL (p. 192)** (p. 192), **ESEL (p. 661)** (p. 661), **KSEL (p. 995)** (p. 995), **LSEL (p. 1074)** (p. 1074), **NSEL (p. 1266)** (p. 1266), and **VSEL (p. 2163)** (p. 2163)).

The default tolerance logic is based on the relative values of VMIN and VMAX as follows:

- If VMIN = VMAX, *Toler* = 0.005 x VMIN.
- If VMIN = VMAX = 0.0, *Toler* = 1.0E-6.
- If VMAX ≠ VMIN, *Toler* = 1.0E-8 x (VMAX-VMIN).

This command is typically used when VMAX-VMIN is very large so that the computed default tolerance is therefore large and the xSEL commands selects more than what is desired.

Toler remains active until respecified by a subsequent **SELTOL (p. 1710)** command. A **SELTOL (p. 1710)** < blank > resets back to the default *Toler* logic.

Menu Paths

This command cannot be accessed from a menu.

SEMIIMPLICIT, *Option*, *Type*, *Value*

Specifies parameters for a semi-implicit solution.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Option to be performed:

ETOI

Criterion for transitioning from the semi-implicit solution phase to the implicit solution phase.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MSCA

Selective mass scaling factor used during the semi-implicit solution phase.

SFAC

Safety factor for time incrementation used during the semi-implicit solution phase.

AUTS

Automatic time stepping and bisection controls used during the semi-implicit solution phase.

BVIS

Bulk viscosity controls used during the semi-implicit solution phase.

EFRQ

Output and restart file frequency used during the semi-implicit solution phase.

Type

Additional input; varies depending on the *Option* value. See table below.

Value

Additional input; varies depending on the *Option* and *Type* values. See table below.

Option **Label**	***Type*** **=**	***Value*** **=**
ETOI	TIME - Time to spend in the semi-implicit phase (default)	Time value (defaults to the minimum time increment specified in the implicit run)
MSCA	DTIM - Initial minimum stable time increment	Time increment; defaults to (1e-4)*(total step time) [1]
	MASS - Selective mass scaling factor	Scaling factor
SFAC	(not used)	Safety factor (defaults to 0.9)
AUTS	DSPL - Automatic time stepping and bisections control based on displacement (default for all analysis types except static)	Maximum allowable jump in the maximum displacement increment between two substeps at which bisection will be triggered (defaults to 100)
	ENER - Automatic time stepping and bisections control based on energy (default for static analysis)	Maximum allowable ratio of kinetic energy to potential energy at which bisectionn will be triggered (defaults to 100). For bisection to trigger, the potential energy

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option Label	*Type* =	*Value* =
		should be more than the program calculated reference value
BVIS	LBVD - Linear bulk viscosity damping factor	Damping factor (defaults to 1.5)
	QBVD - Quadratic bulk viscosity damping factor	Damping factor (defaults to 0.06)
	RBVD - Rotational bulk viscosity damping factor	Damping factor (defaults to 0.06)
	LIMI - Bulk viscosity force upper limit	Upper limit percentage (general default is 0.05; for PLANE182 and SOLID185 with reduced integration, the default is 0.01)
	NONE - Bulk viscosity is not included	(not used)
EFRQ	OUTR - Solution output frequency (overwrites *Freq* value on **OUTRES (p. 1336)**). **SEMIIMPLICIT (p. 1710)**,EFRQ,OUTR must be issued after **OUTRES (p. 1336)**.	Frequency value (defaults to 100) [2]
	RESC - Frequency for writing restart files (overwrites *Frequency* value on **RESCONTROL (p. 1592)**). **SEMIIMPLICIT (p. 1710)**,EFRQ,RESC must be issued after **RESCONTROL (p. 1592)**.	Frequency value (defaults to 100) [2]

1. The default time increment (1e-4)*(total step time) is based on the total step time for the load step in which the solution transitions to the semi-implicit phase.

2. The frequencies specified by *Option* = EFRQ are based on *all* substeps, starting from the beginning of the analysis and spanning both the implicit and semi-implicit solution phases. For example, during the semi-implicit solution phase results are written for any substep that is divisible by the specified OUTR value.

Command Default

By default, the semi-implicit method is not used.

If the **SEMIIMPLICIT (p. 1710)** command is issued with no arguments specified, the semi-implicit scheme is activated with the following defaults:

The safety factor for time incrementation = 0.9.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selective mass scaling factor is set such that the initial stable time increment in the semi-implicit phase is (1e-4)*(total step time).

For most analysis types, bisection is triggered if the maximum displacement increment increases more than 100 times that of the previous substep.

In a static analysis, bisection is triggered if the kinetic energy is more than 100 times the potential energy and the potential energy is more than the program calculated reference value.

Bulk viscosity is included.

The frequency at which the results file and restart files are written during the semi-implicit phase is every 100 substeps.

Notes

This command triggers a semi-implicit solution scheme in which the analysis transitions to a semi-implicit solution method when the implicit solution method fails to converge. The command is valid only in the solution processor (**/SOLU (p. 1821)**) and must be defined before the first **SOLVE (p. 1822)** command.

The **SEMIIMPLICIT (p. 1710)** command can be used in a restart, even if the base analysis did not include the command. Therefore, a problem that failed in the implicit analysis can be restarted with this command so that it can transition to the semi-implicit method and solve further.

The **SEMIIMPLICIT (p. 1710)** command can overwrite the values on some commands, as described in the following table.

Commands Affected by **SEMIIMPLICIT (p. 1710)**	Behavior
NSUBST (p. 1287) and **DELTIM (p. 515)**	During the semi-implicit solution phase, the time incrementation does not follow that set by the **NSUBST (p. 1287)** and **DELTIM (p. 515)** commands.
ETCONTROL (p. 701)	If the **SEMIIMPLICIT (p. 1710)** command is issued, the **ETCONTROL (p. 701)** command is ignored.
OUTRES (p. 1336)	During the semi-implicit solution phase, the frequency for writing results specified by the **OUTRES (p. 1336)** command is ignored, unless a negative number was specified on **OUTRES (p. 1336)**.
RESCONTROL (p. 1592)	During the semi-implicit solution phase, the restart file writing frequency specified by the **RESCONTROL (p. 1592)** command is ignored, unless a negative number was specified on **RESCONTROL (p. 1592)**.
AUTOTS (p. 207)	During the semi-implicit solution phase, automatic time incrementation is on, irrespective of what was specified on **AUTOTS (p. 207)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CUTCONTROL (p. 445)	During the semi-implicit solution phase, the **CUTCONTROL (p. 445)** command is ignored.

Menu Paths

This command cannot be accessed from a menu.

SENERGY, *OPT*, *ANTYPE*

Determines the stored magnetic energy or co-energy.

POST1 (p. 51): Magnetics Calculations (p. 55)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

OPT

Item to be calculated:

0

Stored magnetic energy.

1

Stored magnetic co-energy.

ANTYPE

Analysis type:

0

Static or transient.

1

Harmonic.

Notes

SENERGY (p. 1714) invokes a Mechanical APDL macro which calculates the stored magnetic energy or co-energy for all selected elements. (For a harmonic analysis, the macro calculates a time-averaged (rms) stored energy.)

A summary table listing the energy by material number is generated. The energy density is also calculated and stored on a per-element basis in the element table (**ETABLE (p. 687)**) with the label MG_ENG (energy density) or MG_COENG (co-energy density). The macro erases all other items in the element table and retains only the energy density or co-energy density.

Issue **PLETAB (p. 1411)** and **PRETAB (p. 1490)** to plot and list the energy density.

The macro is valid for static and low-frequency magnetic field formulations.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The macro will not calculate stored energy and co-energy for the following cases:

- Orthotropic nonlinear permanent magnets.
- Orthotropic nonlinear permeable materials.
- Temperature dependent materials.

SENERGY (p. 1714) is restricted to MKSA units.

Menu Paths

Main Menu>General Postproc>Elec&Mag Calc>Element Based>Co-Energy
Main Menu>General Postproc>Elec&Mag Calc>Element Based>Energy

---

SEOPT,*Sename*,*SEMATR*,*SEPR*,*SESST*,*ExpMth*,*SeOcLvL*,--,*LPnameKey*,*SeEdgeKey*

Specifies substructure analysis options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sename

The name (case-sensitive, 32-character maximum) assigned to the .sub superelement matrix file. If not specified, the name defaults to *Fname* on the **/FILNAME (p. 757)** command.

SEMATR

Matrix generation key:

1

Generate stiffness (or conductivity) matrix (default).

2

Generate stiffness and mass (or conductivity and specific heat) matrices.

3

Generate stiffness, mass and damping matrices.

SEPR

Print key:

0

Do not print superelement matrices or load vectors (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Print both load vectors and superelement matrices.

2

Print load vectors but not matrices.

SESST

Stress-stiffening key:

0

Do not save space for stress stiffening in a later run.

1

Save space for the stress stiffening matrix (calculated in a subsequent generation run after the expansion pass).

ExpMth

Expansion method for expansion pass:

BACKSUB

Save necessary factorized matrix files (for example, the .LN22 file) for backsubstitution during the subsequent expansion passes (default). This normally results in a large usage of disk space.

MODDIR

This is the same expansion method as BACKSUB, except that the static correction vectors (see the first term of Equation 15.134 in the *Mechanical APDL Theory Reference*) are stored on the .bclv file instead of the .LN22 file. This option is required when remote read-only file usage is used during the first solution of the first restart of a generation pass (see the **MODDIR (p. 1134)** command).

RESOLVE

Do not save factorized matrix files. Global stiffness matrix will be reformed during expansion pass. This option provides an effective way to save disk space usage. This option cannot be used if the use pass uses large deflections (**NLGEOM (p. 1231)**,ON).

NONE

Do not save factorized matrix files. With this option, the expansion pass is not possible when factorized matrix files are required.

BCLV

Do not save factorized matrix files. The static correction vectors (see the first term of Equation 15.134 in the *Theory Reference*) are stored in the .bclv file. With this option, the expansion pass is not possible when factorized matrix files are required.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SeOcLvL

For the added-mass calculation, the ocean level to use when ocean waves (**OCTYPE (p. 1320)**,,WAVE) are present:

ATP

The ocean level at this point in time (default).

MSL

The mean ocean level.

- -

Unused field.

LPnameKey

ON

All files created during the generation pass are named using *Sename*.

OFF

All files created during the generation pass are named using the Jobname defined for the analysis (see **/FILNAME (p. 757)** command) except for the .sub file, which uses *Sename* if specified (default).

SeEdgeKey

Key to control confidentiality of the shape of the substructure finite element model:

ON

Substructure edges are written to the .sub file and can be viewed as an edge outline in graphics displays in the use pass by issuing **EPLOT (p. 648)** (default).

OFF

Substructure edges are not written to the .sub file and cannot be viewed as an edge outline in graphics displays in the use pass.

Notes

SEOPT (p. 1715) specifies substructure analysis options (**ANTYPE (p. 162)**,SUBSTR). If used during solution, the command is valid only within the first load step.

When ocean waves (**OCTYPE (p. 1320)**,,WAVE) are present, the *SeOcLvL* argument specifies the ocean height or level to use for the added-mass calculation, as the use-run analysis type is unknown during the generation run.

The expansion pass method RESOLVE is not supported with component mode synthesis analysis (**CMSOPT (p. 378)**). *ExpMth* is automatically set to BACKSUB for CMS analysis. The RESOLVE method inval-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

idates the use of the **NUMEXP (p. 1293)** command. The RESOLVE method does not allow the restarting of the substructure generation pass and the computation of results based on nodal velocity and nodal acceleration (damping force, inertial force, kinetic energy, etc.) in the substructure expansion pass.

If *ExpMth* = NONE or BCLV in a substructure analysis or component mode synthesis (CMS) analysis (**CMSOPT (p. 378)**) using the fixed-interface (**CMSOPT (p. 378)**,FIX) or free-interface (**CMSOPT (p. 378)**,FREE) methods, you cannot restart the generation pass or perform the expansion pass. (In CMS analyses using those methods, however, the expansion pass is possible when element-results calculation is activated in the generation pass (*Elcalc* = YES on **CMSOPT (p. 378)**)). The *ExpMth* argument is not required for CMS analysis using the residual-flexible free-interface method (**CMSOPT (p. 378)**,RFFB).

For linear perturbation substructure generation pass with multiple substructures, set *LPnameKey* = ON to avoid copying files at the end of each generation pass with the **/COPY (p. 416)** command. For an example with CMS analysis using the fixed-interface method, see Linear Perturbation Substructure with Two Generation Passes, Use Pass, and Two Expansion Passes in the *Structural Analysis Guide*.

To verify superelement location as an edge outline in graphics displays in the use pass by issuing **EPLOT (p. 648)**, *SeEdgeKey* must be set to ON. To keep the shape of the substructure finite element model confidential and prohibit visualization of the superelement and any transformed superelements created with the **SETRAN (p. 1730)** or **SESYMM (p. 1718)** commands, set *SeEdgeKey* to OFF.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

SESYMM,*Sename*,*Ncomp*,*INC*,*File*,*Ext*,--

Performs a symmetry operation on a superelement within the use pass.

PREP7 (p. 22): Superelements (p. 34)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sename

The name (case-sensitive) of the superelement matrix file created by the substructure generation pass (*Sename*.sub). Defaults to the current Jobname. If a number, it is the element number of a previously defined superelement in the current use pass.

Ncomp

Symmetry key:

X

X symmetry (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y

Y symmetry.

Z

Z symmetry.

INC

Increment all nodes in the superelement by *INC*.

File

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

This field must be input.

Ext

Filename extension (eight-character maximum).

The extension defaults to SUB.

--

Unused field.

Notes

Performs a symmetry operation on a superelement within the substructure use pass by reversing the sign of component *Ncomp* in the global Cartesian coordinate system. The node numbers are incremented by *INC*. The new superelement is written to `File.sub` in the current directory (by default). All master node nodal coordinate systems must be global Cartesian (no rotated nodes allowed).

The maximum number of transformations for a given superelement is five (including **SETRAN (p. 1730)**, **SESYMM (p. 1718)**, and the large rotation transformation if **NLGEOM (p. 1231)** is ON in the use pass).

This command is not supported if the original superelement matrix was created in a component mode synthesis analysis generation pass with the element results calculation activated (*Elcalc* = YES on **CMSOPT (p. 378)**).

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By Reflection

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*SET, *Par*, *VALUE*, *VAL2*, *VAL3*, *VAL4*, *VAL5*, *VAL6*, *VAL7*, *VAL8*, *VAL9*, *VAL10*

Assigns values to user-named parameters.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

An alphanumeric name used to identify this parameter. *Par* can contain up to 32 characters, beginning with a letter and containing only letters, numbers, and underscores. *Examples:* **ABC A3X TOP_END**

Command names, function names, label names, component and assembly names, etc., are invalid, as are parameter names beginning with an underscore (for example, **_LOOP**).

Parameter names ending in an underscore are not listed by the ***STATUS (p. 1863)** command. Array parameter names must be followed by a subscript, and the entire expression must be 32 characters or less. *Examples:* **A(1,1) NEW_VAL(3,2,5) RESULT(1000)**

There is no character parameter substitution for the *Par* field. Table parameters used in command fields (where constant values are normally given) are limited to 32 characters.

VALUE

Numerical value or alphanumeric character string (up to 32 characters enclosed in single quotes) to be assigned to this parameter. Examples: **A(1,3)=7.4 B='ABC3'** Can also be a parameter or a parametric expression. *Examples:* **C=A(1,3) A(2,2)=(C+4)/2**. If *VALUE* is the table array name, the subscripts are the values of the primary variables and the table is evaluated at these specified index values.

If blank, delete this parameter. *Example:* **A=** deletes parameter **A**.

VAL2, VAL3, VAL4, VAL5, VAL6, VAL7, VAL8, VAL9, VAL10

If *Par* is an array parameter, values *VAL2* through *VAL10* (up to the last nonblank value) are sequentially assigned to the succeeding array elements of the column. *Examples:*
***SET,A(1,4),10,11** assigns **A(1,4)=10, A(2,4)=11**, and
***SET,B(2,3),'file10','file11'** assigns **B(2,3)='file10', B(3,3)='file11'**.

Notes

Assigns values to user-named parameters that can be substituted later in the run. The equivalent (and recommended) format is

Par = *VALUE*,*VAL2*,*VAL3*, . . . , *VAL10*

which can be used in place of ***SET (p. 1720)**,*Par*, ... for convenience.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter Definitions

Parameters (numeric or character) can be scalars (single valued) or arrays (multiple valued in one, two, or three dimensions). An unlimited number of parameter names can be defined in any run. For very large numbers of parameters, it is most efficient to define them in alphabetical order.

Parameter values can be redefined at any time. Array parameters can also be assigned values within a do-loop (***DO (p. 567)**) for convenience. Internally programmed do-loop commands are also available with the ***VXX*** commands (***VFILL (p. 2097)**). Parameter values (except for parameters ending in an underscore) can be listed with the ***STATUS (p. 1863)** command, displayed via ***VPLOT (p. 2146)** (numeric parameters only), and modified via ***VEDIT (p. 2089)** (numeric parameters only).

Older program-provided macro files can use parameter names that do not begin with an underscore. Using these macros embedded in your own macros may cause conflicts if the same parameter names are used.

Parameters can also be resolved in comments created via **/COM (p. 404)**. A parameter can be deleted by redefining it with a blank *VALUE*. If the parameter is an array, the entire array is deleted. Parameters can also be defined by a response to a query via ***ASK (p. 84)** or from a Mechanical APDL-provided value via ***GET (p. 797)**.

Array Parameters

Array parameters must be dimensioned (***DIM (p. 530)**) before being assigned values unless they are the result of an array operation or defined using the implied loop convention.

Undefined scalar parameters are initialized to a near-zero value. Numeric array parameters are initialized to zero when dimensioned, and character array parameters are initialized to blank.

An existing array parameter must be deleted before it can be redimensioned.

Array parameter names must be followed by a subscript list (enclosed in parentheses) identifying the element of the array. The subscript list can have one, two, or three values (separated by commas). Typical array parameter elements are A(1,1), NEW_VAL(3,2,5), RESULT(1000). Subscripts for defining an array element must be integers (or parameter expressions that evaluate to integers). Non-integer values are rounded to the nearest integer value.

All array parameters are stored as 3D arrays with the unspecified dimensions set to 1. For example, the 4th array element of a 1-dimensional array, A(4), is stored as array element A(4,1,1).

Arrays adhere to standard FORTRAN conventions.

Numerical Parameter Substitution

If the parameter name *Par* is input in a numeric argument of a command, the numeric value of the parameter (as assigned with ***SET (p. 1720)**, ***GET (p. 797)**, =, etc.) is substituted into the command at that point. Substitution occurs only if the parameter name is used between blanks, commas, parentheses, or arithmetic operators (or any combination) in a numeric argument. Substitution can be prevented by enclosing the parameter name *Par* within single quotes ('), if the parameter is alone in the argument; if the parameter is part of an arithmetic expression, the entire expression must be enclosed within single quotes to prevent substitution. In either case the character string will be used instead of the numeric value (and the string will be taken as 0.0 if it is in a numeric argument).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A forced substitution is available in the text fields of **/TITLE (p. 2007)**, **/STITLE (p. 1867)**, **/TLABEL (p. 2007)**, **/AN3D (p. 129)**, **/SYP (p. 1894)** (*ARG1*--*ARG8*), and ***ABBR (p. 81)** by enclosing the parameter within percent (%) signs. Also, parameter substitution can be forced within the file name or extension fields of commands having these fields by enclosing the parameter within percent (%) signs. Array parameters (***DIM (p. 530)**) must include a subscript (within parentheses) to identify the array element whose value is to be substituted, such as A(1,3). Out-of-range subscripts result in an error message. Non-integer subscripts are allowed when identifying a TABLE array element for substitution. A proportional linear interpolation of values among the nearest array elements is performed before substitution. Interpolation is done in all three dimensions.

Note:

Interpolation is based upon the assigned index numbers which must be defined when the table is filled (***DIM (p. 530)**).

Character Parameter Substitution

Most alphanumeric arguments permit the use of character parameter substitution. When the parameter name *Par* input, the alphanumeric value of the parameter is substituted into the command at that point. Substitution can be suppressed by enclosing the parameter name within single quotes ('). Forced substitution is available in some fields by enclosing the parameter name within percent (%) signs. Valid forced substitution fields include command name fields, *Fname* (filename) or *Ext* (extension) arguments, ***ABBR (p. 81)** command (*Abbr* arguments), **/TITLE (p. 2007)** and **/STITLE (p. 1867)** commands (*Title* argument) and **/TLABEL (p. 2007)** command (*Text* argument). Character parameter substitution is also available in the ***ASK (p. 84)**, **/AN3D (p. 129)**, ***CFWRITE (p. 318)**, ***IF (p. 927)**, ***ELSEIF (p. 620)**, ***MSG (p. 1185)**, ***SET (p. 1720)**, ***USE (p. 2059)**, ***VREAD (p. 2153)**, and ***VWRITE (p. 2174)** commands. Character array parameters must include a subscript (within parentheses) to identify the array element whose value is to be substituted.

Parameter Expressions

If a parameter *operation expression* is input in a numeric argument, the numeric value of the expression is substituted into the command at that point. Allowable operation expressions are of the form

E1oE2oE3 ...oE10

where E1, E2, etc. are expressions connected by operators (o). The allowable operations (o) are

+ - * / ** < >

For example, **A+B**C/D*E** is a valid operation expression. The ***** represents multiplication and the ****** represents exponentiation.

Note:

Exponentiation of a negative number (without parentheses) to an integer power follows standard FORTRAN hierarchy conventions; that is, the positive number is exponentiated and then the sign is attached. Thus, -4**2 is evaluated as -16. If parentheses are applied, such as (-4)**2, the result is 16.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A parameter is evaluated as a number within parentheses before exponentiation. Exponentiation of a negative number to a non-integer power is performed by exponentiating the positive number and prepending the minus sign, for example, -4**2.3 is -(4**2.3). The < and > operators allow conditional substitution. For example, E1<E2 substitutes the value of E1 if the comparison is true or the value of E2 if the comparison is false.

Do not use spaces around operation symbols, as " *" (a space and a star) makes the remainder of the line a comment. Operation symbols (or symbols and signs) cannot be immediately adjacent to each other. Parentheses can be used to separate symbols and signs, to determine a hierarchy of operations, or for clarity. For example, use A**(-B) instead of A**-B. Numbers ending with +0nn or -0nn are assumed to be of exponential form (as written on files by some computer systems) so that 123-002 is 123E-2 while 123-2 is 121. Avoid inputting this form of exponential data directly. The default hierarchy follows the standard FORTRAN conventions, namely:

- operations in parentheses (innermost first)
- then exponentiation (right to left)
- then multiplication or division (left to right)
- then unary association (such as +A or -A)
- then addition or subtraction (left to right)
- then logical evaluations (left to right).

Expressions (E) can be a constant, a parameter, a function, or another operation expression (of the form E1oE2oE3 ...oE10). Functions are of the form FTN(A) where the argument (A) can itself be of the form E1oE2oE3 ...oE10. Operations are recursive to a level of four deep (three levels of internally nested parentheses). Iterative floating point parameter arithmetic should not be used for high precision input because of the accumulated numerical round off-error. Up to 10 expressions are accepted within a set of parenthesis.

Valid functions (which are based on standard FORTRAN functions where possible) are:

SIN(*X*)	Sine
COS(*X*)	Cosine
TAN(*X*)	Tangent
ASIN(*X*)	Arcsine
ACOS(*X*)	Arccosine
ATAN(*X*)	Arctangent
ATAN2(*Y*,*X*)	Arctangent (*Y*/*X*) with the sign of each component considered
SINH(*X*)	Hyperbolic sine
COSH(*X*)	Hyperbolic cosine
TANH(*X*)	Hyperbolic tangent
SQRT(*X*)	Square root
ABS(*X*)	Absolute value
SIGN(*X*,*Y*)	Absolute value of *X* with sign of *Y*. *Y*=0 results in positive sign
NINT(*X*)	Nearest integer

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MOD(*X*,*Y*)	Remainder of *X*/*Y* *Y*=0 returns zero (0)
EXP(*X*)	Exponential
LOG(*X*)	Natural log
LOG10(*X*)	Common log
RAND(*X*,*Y*)	Random number, where *X* is the lower bound, and *Y* is the upper bound
GDIS(*X*,*Y*)	Random sample of Gaussian distributions, where *X* is the mean, and *Y* is the standard deviation
LWCASE(*CPARM*)	Lowercase equivalent of character parameter *CPARM*
UPCASE(*CPARM*)	Uppercase equivalent of character parameter *CPARM*
VALCHR(*CPARM*)	Numeric value of character parameter *CPARM* (If *CPARM* is a numeric parameter, returns 0.0)
CHRVAL(*PARM*)	Character value of numerical parameter *PARM*. For ABS(*PARM*) < 10, character value format is F8.5; for 10 ≤ ABS(*PARM*) < 1000, format is F8.3; for 1,000 ≤ ABS(*PARM*)< 10,000,000, format is F8.0. For 10,000,000 ≤ PARM < 100,000,000, format is also F8.0. Otherwise result is 0.0 and is not a character value.
IBSET(b1,n2)	Set the n2 bit in value b1 (bits are numbered from 0 to 31)
IBCLR(b1,n2)	Clear the n2 bit in value b1
BTEST(b1,n2)	Test the n2 bit in value b1 (return true (1.0) if bit is set)
BITAND(b1,b2)	Bitwise AND of value b1 and b2
BITOR(b1,b2)	Bitwise OR of value b1 and b2
BITXOR(b1,b2)	Bitwise XOR of value b1 and b2
BITSET(b1,b2)	Set the b2 bits in b1
BITCLEAR(b1,b2)	Clear the b2 bits in b1

Function arguments (*X*,*Y*, etc.) must be enclosed within parentheses and can be numeric values, parameters, or expressions. Input arguments for angular functions must evaluate to radians by default. Output from angular functions are also in radians by default. See ***AFUN (p. 83)** to use degrees instead of radians for the angular functions. See ***VFUN (p. 2106)** for applying these parameter functions to a sequence of array elements. Additional functions, called *get functions*, are described via ***GET (p. 797)**.

Menu Paths

Utility Menu>Parameters>Scalar Parameters

SET,*Lstep*,*Sbstep*,*Fact*,*KIMG*,*TIME*,*ANGLE*,*NSET*,*ORDER*

Defines the data set to be read from the results file.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lstep

Load step number of the data set to be read (defaults to 1):

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N

Read load step *N*.

FIRST

Read the first data set (*Sbstep* and *TIME* are ignored).

LAST

Read the last data set (*Sbstep* and *TIME* are ignored).

NEXT

Read the next data set (*Sbstep* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

PREVIOUS

Read the previous data set (*Sbstep* and *TIME* are ignored). If at the first data set, the last data set will be read as the previous.

NEAR

Read the data set nearest to *TIME* (*Sbstep* is ignored). If *TIME* is blank, read the first data set.

LIST

Scan the results file and list a summary of each load step. (*KIMG*, *TIME*, *ANGLE*, and *NSET* are ignored.)

Sbstep

Substep number (within *Lstep*). Defaults to the last substep of the load step (except in a buckling or modal analysis). For a buckling (**ANTYPE (p. 162)**,BUCKLE) or modal (**ANTYPE (p. 162)**,MODAL) analysis, *Sbstep* corresponds to the mode number. Specify *Sbstep* = LAST to store the last substep for the specified load step (that is, issue a **SET (p. 1724)**,*Lstep*,LAST command).

If *Lstep* = LIST, *Sbstep* = 0 or 1 lists the basic step information. *Sbstep* = 2 also lists the basic step information, but includes the load step title, and labels imaginary data sets if they exist.

Fact

Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. A nonzero factor excludes non-summable items.

Harmonic velocities or accelerations can be calculated from the displacement results from a modal (**ANTYPE (p. 162)**,MODAL) or harmonic (**ANTYPE (p. 162)**,HARMIC) analysis. If *Fact* = VELO, the harmonic velocities (v) are calculated from the displacements (d) at a particular frequency (f) according to the relationship $v = 2 \pi f d$. Similarly, if *Fact* = ACEL, the harmonic accelerations (a) are calculated as $a = (2 \pi f)^2 d$.

If *Lstep* = LIST in an analysis using rezoning, *Fact* across all rezoning data sets is listed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KIMG

Used only with complex results (harmonic and complex modal analyses).

0 or REAL

Store the real part of complex solution (default).

1, 2 or IMAG

Store the imaginary part of a complex solution.

3 or AMPL

Store the amplitude.

4 or PHAS

Store the phase angle. The angle value, expressed in degrees, will be between -180° and +180°.

TIME

Time-point identifying the data set to be read. For a harmonic analyses, time corresponds to the frequency.

For a buckling analysis, time corresponds to the load factor.

Used only in the following cases: If *Lstep* = NEAR, read the data set nearest to *TIME*. If both *Lstep* and *Sbstep* are zero (or blank), read data set at time = *TIME*.

Do not use *TIME* to identify the data set to be read if you used the arc-length method (**ARCLEN (p. 175)**) in your solution.

If *TIME* is between two solution time points in the results file, a linear interpolation occurs between the two data sets (except in cases where rezoning or selected results output (**OSRESULT (p. 1327)**) is used).

Solution items not written to the results file (**OUTRES (p. 1336)**) for either data set will result in a null item after data set interpolation.

If *TIME* is beyond the last time point on the file, the last time point will be used.

If *TIME* is between two solution time points and both *Lstep* and *Sbstep* are zero (or blank), no interpolation is performed for the **PRCINT (p. 1472)**/**PLCINT (p. 1396)** commands. (That is, for results generated by the **CINT (p. 334)** command, only the data set associated with the lower of the solution time points is used.)

ANGLE

For harmonic elements (PLANE25, PLANE75, PLANE78, PLANE83, and SHELL61), *ANGLE* specifies the circumferential location (0.0 to 360°) used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the displacements and element results, and to

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the applied constraints and loads which overwrites any values existing in the database. If *ANGLE* = NONE, all harmonic factors are set to 1 and postprocessing yields the solution output.

When using *ANGLE* = NONE with *MODE* > 0, the combined stresses and strains are not valid.

The default value of *ANGLE* is 0.0; however if the **SET (p. 1724)** command is not used, the effective default is NONE.

For full harmonic analyses with the amplitude option (*KIMG* = 3 or AMPL), *ANGLE* is the prescribed phase angle at which the amplitude is computed.

NSET

Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *Sbstep*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **SET (p. 1724)**,LIST.

ORDER

Key to sort the harmonic index results. This option applies to cyclic symmetry buckling and modal analyses only, and is valid only when *Lstep* = FIRST, LAST, NEXT, PREVIOUS, NEAR or LIST.

ORDER

Sort the harmonic index results in ascending order of eigenfrequencies or buckling load multipliers.

(blank)

No sorting takes place.

Notes

Defines the data set to be read from the results file into the database. Various operations may also be performed during the read operation. The database must have the model geometry available (or use the **RESUME (p. 1601)** command before the **SET (p. 1724)** command to restore the geometry from Jobname.db). Values for applied constraints (**D (p. 483)**) and loads (**F (p. 729)**) in the database will be replaced by their corresponding values on the results file, if available. (See the description of the **OUTRES (p. 1336)** command.) In a single load step analysis, these values are usually the same, except for results from harmonic elements. (See the description of the *ANGLE* value above.)

In an interactive run, the sorted list (*ORDER* option) is also available for results-set reading via a GUI pick option.

You can postprocess results without issuing a **SET (p. 1724)** command if the solution results were saved to the database file (Jobname.db). A distributed-memory parallel simulation, however, can only postprocess using the results file (for example, Jobname.rst) and cannot use the Jobname.db file since no solution results are written to the database. Therefore, you must issue a **SET (p. 1724)** command or a **RESCOMBINE (p. 1589)** command before postprocessing in a distributed-memory parallel run.

When postprocessing amplitudes or phases (*KIMG* = AMPL or PHAS):

- The only appropriate coordinate system is the solution coordinate system (**RSYS (p. 1639)** ,SOLU). For layered elements, a layer (**LAYER (p. 1015)**) must also be specified. When displaying the displacement

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

amplitudes, use a contour display (**PLNSOL (p. 1425)** command). Because a deformed shape display (**PLDISP (p. 1400)** command) could lead to a non-physical shape, the displacement scaling is off by default (**/DSCALE (p. 575)**,,OFF).

- The conversion is not valid for averaged results, derived results (such as principal stress/strain, equivalent stress/strain, and USUM), or summed results obtained using **FSUM (p. 780)**, **NFORCE (p. 1216)**, **PRNLD (p. 1503)**, and **PRRFOR (p. 1517)**.
- Cyclic symmetry and multistage cyclic symmetry analysis results are not supported. To compute the amplitude of a single complex mode that was calculated in a cyclic symmetry or multistage cyclic symmetry modal analysis using the damped eigensolver (**MODOPT (p. 1140)**, DAMP), issue the **MAXCYCMODE (p. 1112)** command.
- Coupled-field analysis results are supported when *KIMG* = AMPL, but they are not supported when *KIMG* = PHAS.

Issuing EXPAND (p. 711) in a Cyclic Symmetry Modal Analysis

After issuing **EXPAND (p. 711)**,*Nrepeat* in a cyclic symmetry modal analysis, subsequent **SET (p. 1724)** commands read data from the results file and expand them to *Nrepeat* sectors.

Issuing MSOPT (p. 1195),EXPAND in a Multistage Cyclic Symmetry Analysis

When issuing **MSOPT (p. 1195)**,EXPAND in a multistage cyclic symmetry analysis, subsequent **SET (p. 1724)** commands read the data set from the specified results file and expand the nodes and elements to the stages and sectors specified via **MSOPT (p. 1195)**,EXPAND.

Menu Paths

Main Menu>General Postproc>List Results>Detailed Summary

Main Menu>General Postproc>List Results>Detailed Summary (Freq ordered)

Main Menu>General Postproc>Read Results>By Load Step

Main Menu>General Postproc>Read Results>By Pick

Main Menu>General Postproc>Read Results>By Pick (Freq ordered)

Main Menu>General Postproc>Read Results>First Set

Main Menu>General Postproc>Read Results>Last Set

Main Menu>General Postproc>Read Results>Next Set

Main Menu>General Postproc>Read Results>Previous Set

Main Menu>General Postproc>Results Summary

Main Menu>General Postproc>Results Summary (Freq ordered)

Utility Menu>List>Results>Load Step Summary

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SETFGAP, *GAP*, *ROPT*, - - , *PAMB*, *ACF1*, *ACF2*, *PREF*, *MFP*

Updates or defines the real constant table for squeeze film elements.

PREP7 (p. 22): Real Constants (p. 23)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

GAP

Gap separation.

ROPT

Real constant set option.

0

Creates separate real constant sets for each selected element with the specified real constant values (default).

1

Updates existing real constant sets. The gap separation is updated from displacement results in the database. Other real constants are updated as specified in the command input parameters.

- -

Unused field

PAMB

Ambient pressure.

ACF1, ACF2

Accommodation factor 1 and 2.

PREF

Reference pressure for mean free path.

MFP

Mean free path.

Notes

This command is used for large signal cases to update the gap separation real constant on a per-element basis. Issue this command prior to solution using the default *ROPT* value to initialize real constant sets for every fluid element. After a solution, you can re-issue the command to update the real constant set for a subsequent analysis. See Introduction for more information on thin film analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Real Constants>ThinFilm

Main Menu>Preprocessor>Real Constants>ThinFilm

Main Menu>Solution>Load Step Opts>Other>Real Constants>ThinFilm

---

SETRAN, *Sename*, *KCNTO*, *INC*, *File*, *Ext*, --, *DX*, *DY*, *DZ*, *NOROT*

Creates a superelement from an existing superelement.

PREP7 (p. 22): Superelements (p. 34)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Sename

The name (case-sensitive) of the file containing the original superelement matrix created by the generation pass (*Sename*.SUB). The default is the current Jobname. If *Sename* is a number, it is the element number of a previously defined superelement in the current use pass.

KCNTO

The reference number of the coordinate system to where the superelement is to be transferred. The default is the global Cartesian system. Transfer occurs from the active coordinate system.

INC

The node offset. The default is zero. All new element node numbers are offset from those on the original by *INC*.

File

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

This field requires input.

Ext

Filename extension (eight-character maximum).

The extension defaults to SUB.

--

Unused field.

DX, DY, DZ

Node location increments in the global Cartesian coordinate system. Defaults to zero.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NOROT

Node rotation key (p. 1731):

0

The nodal coordinate systems of the transferred superelement rotate into the *KCNTO* system. (That is, the nodal coordinate systems rotate with the superelement.) The superelement matrices remain unchanged. This value is the default.

1

The nodal coordinate systems do not rotate. (That is, they remain fixed in their original global orientation.) The superelement matrices and load vectors are modified if any rotations occur.

Notes

The **SETRAN (p. 1730)** command creates a superelement from an existing superelement and writes the new element to a file. You can then issue an **SE (p. 1651)** command to read the new element (during the use pass).

You can create a superelement from an original by:

- Transferring the original's geometry from the active coordinate system into another coordinate system (*KCNTO*)
- Offsetting its geometry in the global Cartesian coordinate system (*DX*, *DY*, and *DZ*)
- Offsetting its node numbers (*INC*).

A combination of methods is valid. If you specify both the geometry transfer and the geometry offset, the transfer occurs first.

If you specify rotation of the transferred superelement's nodal coordinate systems into the *KCNTO* system (*NOROT* = 0), the rotated nodes cannot be coupled via the **CP (p. 421)** command; in this case, issue the **CE (p. 297)** command instead. If you specify *no* rotation of the nodal coordinate systems (*NOROT* = 1) for models with displacement degrees of freedom, and *KCNTO* is not the active system, the superelement *Sename* must have six MDOF at each node that has MDOF; therefore, only elements with all six structural DOFs are valid in such cases.

There is no limit to the number of copies that can be made of a superelement, provided the copies are all generated from the same original superelement. However, nested copies are limited to five. In other words, the total number of different *Sename* usages on the **SETRAN (p. 1730)** and **SESYMM (p. 1718)** commands is limited to five.

This command is not supported if the original superelement matrix was created in a component mode synthesis analysis generation pass with the element results calculation activated (*Elcalc* = YES on **CMSOPT (p. 378)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By CS Transfer

Main Menu>Preprocessor>Modeling>Create>Elements>Superelem>By Geom Offset

---

SEXP, *LabR*, *Lab1*, *Lab2*, *EXP1*, *EXP2*

Forms an element table item by exponentiating and multiplying.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1

First labeled result item in operation.

Lab2

Second labeled result item in operation (may be blank).

EXP1

Exponent applied to *Lab1*.

EXP2

Exponent applied to *Lab2*.

Notes

Forms a labeled result item (see **ETABLE (p. 687)** command) for the selected elements by exponentiating and multiplying two existing labeled result items according to the operation:

$$LabR = (|Lab1|^{EXP1}) \times (|Lab2|^{EXP2})$$

Roots, reciprocals, and divides may also be done with this command.

Menu Paths

Main Menu>General Postproc>Element Table>Exponentiate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SF,*Nlist*,*Lab*,*VALUE*,*VALUE2*,–,*MESHFLAG*

Defines surface loads on nodes.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1738)

Nlist

Nodes defining the surface upon which the load is to be applied. Use the label ALL or P, or a component name. If ALL, all selected nodes (**NSEL (p. 1266)**) are used (default). If P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Lab

Valid surface load label. Load labels are listed under **Surface Loads** in the input table for each element type.

Discipline	Surface Load Label	Label Description
Structural	PRES	Pressure
	FREQ	Frequency (harmonic analyses only)
	MXWF	Equivalent source surface flag
Thermal	CONV [1]	Convection
	HFLUX [1]	Heat flux
	RAD	Radiation
	RDSF	Surface-to-surface radiation
Acoustic fluid	FSI [2]	Fluid-structure interaction flag
	IMPD	Impedance or admittance coefficient
	SHLD	Surface normal velocity or acceleration
	MXWF	Maxwell surface flag or equivalent source surface flag
	FREE	Free surface flag
	INF	Exterior Robin radiation boundary flag
	PORT	Port number
	ATTN	Absorption coefficient and transmission loss
	BLI	Viscous-thermal boundary layer surface flag
	RIGW	Rigid wall flag (Neumann boundary)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Surface Load Label	Label Description
	FSIN	One-way structure-to-acoustic coupling interface number
	PRES	Pressure in viscous-thermal acoustics
	CONV	Heat flux in viscous-thermal acoustics
	VIMP	Viscous impedance in viscous-thermal acoustics
	TIMP	Thermal impedance in viscous-thermal acoustics
	PERM	Permeability in poroelastic acoustics
Magnetic	MXWF	Maxwell force flag
Electric	CHRGS	Surface charge density
	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for INFIN110 and INFIN111
Field-surface interface	FSIN	Field-surface interface number
Poromechanics	FFLX	Fluid flow flux
Diffusion	DFLUX	Diffusion flux

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID29, FLUID30, FLUID220, and FLUID221 elements.

VALUE

Surface load value or table name reference for specifying tabular boundary conditions.

If *Lab* = PRES, *VALUE* is the real component of the pressure.

If *Lab* = CONV:

VALUE is typically the film coefficient and *VALUE2* (below) is typically the bulk temperature. If *VALUE* = -*N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* (**MP (p. 1160)**). (See the **SCOPT (p. 1649)** command for a way to override this option and use -*N* as the film coefficient.) The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements.

If **KBC (p. 970)**,0 has been issued for ramped loads, it affects only *VALUE2* the bulk temperature (p. 1736), and the film coefficient specification is unaffected.

In a viscous-thermal acoustic analysis, if *Lab* = CONV, *VALUE* is the real part of the heat flux and *VALUE2* is the imaginary part of the heat flux.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *Lab* = RAD, *VALUE* is surface emissivity.

If *Lab* = PORT, *VALUE* is a port number representing a waveguide exterior port. The port number must be an integer between 1 and 50. For acoustic 2×2 transfer admittance matrix, the port number can be any positive integer. The smaller port number corresponds to the port 1 of the 2×2 transfer admittance matrix and the greater number corresponds to the port 2. If one port of the transfer admittance matrix is connecting to the acoustic-structural interaction interface, the port number corresponds to the port 2 of the transfer admittance matrix. A pair of ports of the 2×2 transfer admittance matrix must be defined in the same element. In an acoustic analysis, the positive port number defines a transparent port, through which the reflected sound pressure wave propagates to the infinity; the negative port number defines a vibro port that is the structural vibration surface.

If *Lab* = SHLD, *VALUE* is the surface normal velocity in a harmonic analysis or in a transient analysis solved with the velocity potential formulation; *VALUE* is the surface normal acceleration in a transient analysis solved with the pressure formulation.

If *Lab* = IMPD, *VALUE* is resistance in (N)(s)/m^3 if *VALUE* > 0 and is conductance in mho if *VALUE* < 0 for acoustic or harmonic response analyses. In acoustic transient analyses, *VALUE2* is not used.

If *Lab* = RDSF, *VALUE* is the emissivity value; the following conditions apply: If *VALUE* is between 0 and 1, apply a single value to the surface. If *VALUE*= *-N*, the emissivity may be a function of the temperature, and is determined from the EMISS property table for material *N* (**MP (p. 1160)**). The material *N* does not need to correlate with the underlying solid thermal elements.

If *Lab* = FSIN in a one-way structure-to-acoustic coupling, *VALUE* is the surface interface number.

If *Lab* = FSIN in a unidirectional Mechanical APDL to CFX analysis, *VALUE* is not used.

If *Lab* = ATTN, *VALUE* is the absorption coefficient of the surface.

If *Lab* = VIMP , *VALUE* is resistance of viscous impedance in (N)(s)/m^3.

If *Lab* = TIMP , *VALUE* is resistance of thermal impedance in (N)(s)/m^3.

If *Lab* = PERM , *VALUE* is permeability in m^2.

VALUE2

Second surface load value (if any).

If *Lab* = PRES, this value is the imaginary pressure component, used by the following supported elements:

- Surface elements: SURF153, SURF154 and SURF159.
- Solid and solid-shell elements: PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, and SOLID285.
- Shell elements: SHELL181, SHELL281, SHELL208, and SHELL209.
- Coupled-field elements with structural degrees of freedom: PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Supported analysis types in this case are:

- Full harmonic (**HROPT (p. 912)**,FULL)
- Mode-superposition harmonic (**HROPT (p. 912)**,MSUP), if the mode-extraction method is Block Lanczos (**MODOPT (p. 1140)**,LANB), PCG Lanczos (**MODOPT (p. 1140)**,LANPCG), Supernode (**MODOPT (p. 1140)**,SNODE), Subspace (**MODOPT (p. 1140)**,SUBSP), or Unsymmetric (**MODOPT (p. 1140)**,UNSYM)

If *Lab* = CONV:

VALUE2 is the bulk temperature for thermal analyses.

If **KBC (p. 970)**,0 has been issued for ramped loads, the bulk temperature is ramped from the value defined by **TUNIF (p. 2027)** to the value specified by *VALUE2* for the first loadstep. If tabular boundary conditions (p. 1737) are defined, the **KBC (p. 970)** command is ignored and tabular values are used.

For viscous-thermal acoustics *VALUE2* is the imaginary part of heat flux.

If *Lab* = RAD, *VALUE2* is the ambient temperature.

If *Lab* = SHLD, *VALUE2* is the phase angle of the normal surface velocity (defaults to zero) for harmonic response analyses while *VALUE2* is not used for transient analyses in acoustics.

If *Lab* = IMPD, *VALUE2* is reactance in (N)(s)/m^3 if *VALUE* > 0 and is the product of susceptance and angular frequency if *VALUE* < 0 for acoustics.

If *Lab* = RDSF, *VALUE2* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will also occur to ambient. If *VALUE2* is negative radiation direction is reversed and will occur inside the element for the flagged radiation surfaces.

If *Lab* = FSIN in a unidirectional Mechanical APDL to CFX analysis, *VALUE2* is the surface interface number (not available from within the GUI).

If *Lab* = PORT, *VALUE2* is not used.

If *Lab* = ATTN, *VALUE2* is the transmission loss (dB) of the coupled wall in an energy diffusion solution for room acoustics.

If *Lab* = VIMP, *VALUE2* is reactance of viscous impedance in (N)(s)/m^3.

If *Lab* = TIMP, *VALUE2* is reactance of thermal impedance in (N)(s)/m^3.

–

Reserved for future use.

MESHFLAG

Specifies how to apply normal pressure loading on the mesh. Valid in a nonlinear adaptivity analysis when *Lab* = PRES and *Nlist* is a nodal component defined prior to any remeshing activity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 – Pressure loading occurs on the current mesh (default).

1 – Pressure loading occurs on the initial mesh for nonlinear adaptivity.

Notes

Individual nodes cannot be entered for this command. The node list is to identify a surface and the *Nlist* field must contain a sufficient number of nodes to define an element surface. The loads are internally stored on element faces defined by the specified nodes. All nodes on an element face (including midside nodes, if any) must be specified for the face to be used, and the element must be selected.

If all nodes defining a face are shared by an adjacent face of another selected element, the face is not free and will not have a load applied. If more than one element can share the same nodes (for example, a surface element attached to a solid element), select the desired element type before issuing the **SF (p. 1733)** command. The **SF (p. 1733)** command applies only to area and volume elements.

For shell elements, if the specified nodes include face one (which is usually the bottom face) along with other faces (such as edges), only face one is used. Where faces cannot be uniquely determined from the nodes, or where the face does not fully describe the load application, issue **SFE (p. 1761)** instead of **SF (p. 1733)**. A load key of 1 (which is typically the first loading condition on the first face) is used if the face determination is not unique. A uniform load value is applied over the element face.

You can use these related surface-load commands with **SF (p. 1733)**:

SFE (p. 1761) – Defines surface loads on *elements*. You can also use it to apply tapered loads on individual element faces.

SFBEAM (p. 1746) – Applies surface loads to beam elements.

SFCONTROL (p. 1748) – Applies general (normal, tangential, and other) surface loads to supported structural elements (p. 1751).

SFCUM (p. 1758) – Accumulates (adds) surface loads applied via **SF (p. 1733)**.

SFDELE (p. 1760) – Delete loads applied via **SF (p. 1733)**.

SFFUN (p. 1771) – Applies loads from a node-vs.-value function.

SFGRAD (p. 1772) – Applies an alternate tapered load.

Tabular boundary conditions

Tabular boundary conditions (*VALUE* = %*tabname*% and/or *VALUE2* = %*tabname*%) are available for the following surface load labels (*Lab*) only: PRES (real and/or imaginary components), CONV (film coefficient and/or bulk temperature; or heat flux for viscous-thermal acoustics), HFLUX, DFLUX (diffusion flux), RAD (surface emissivity and ambient temperature), IMPD (resistance and reactance), SHLD (normal velocity and phase or acceleration), ATTN (absorption coefficient or transmission loss), VIMP (viscous impedance), and TIMP (thermal impedance). Issue ***DIM (p. 530)** to define a table.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in the PREP7 and **/MAP (p. 1107)** processors.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Product Restrictions

Ansys Mechanical Enterprise

SF (p. 1733),FSI and **SF (p. 1733)**,FSIN are available only in the Ansys Mechanical Enterprise family of products (Ansys Mechanical Enterprise, Ansys Mechanical Enterprise PrepPost, and Ansys Mechanical Enterprise Solver).

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppImped_E>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppShield>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Node Components

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppImped_E>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppShield>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Nodes

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Nodes

Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Nodes

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Nodes

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Nodes

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Nodes

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Node Components

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Nodes

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Nodes

Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Nodes

The following commands cannot be accessed from a menu: SF,,ATTN; SF,,RIGW; SF,,VIMP; SF,,TIMP; and SF,,PERM.

SFA,*Area*,*LKEY*,*Lab*,*VALUE*,*VALUE2*

Specifies surface loads on the selected areas.

SOLUTION (p. 38): Solid Surface Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Area

Area to which surface load applies. If ALL, apply load to all selected areas (**ASEL (p. 192)**). If *Area* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component may be substituted for *Area*.

LKEY

Load key associated with surface load (defaults to 1). Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data table for each element type in the *Element Reference*. *LKEY* is ignored if the area is the face of a volume region meshed with volume elements.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each area type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
	RAD	radiation
	RDSF	surface-to-surface radiation
Acoustic fluid	FSI[2]	fluid-structure interaction flag
	IMPD	impedance boundary flag
	SHLD	surface normal velocity or acceleration
	MXWF	Maxwell surface flag or equivalent source surface
	FREE	free surface flag

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Body Load Label	Label Description
	INF	exterior Robin radiation boundary flag
	PORT	Port number
	ATTN	Absorption coefficient
	BLI	Viscous-thermal boundary layer surface flag
Magnetic	MXWF	Maxwell force flag
Electric	CHRGS	surface charge density
	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for INFIN110 and INFIN111
Field-surface interface	FSIN	field-surface interface number

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements.

VALUE

Surface load value or table name reference for specifying tabular boundary conditions.

If *Lab* = CONV:

VALUE is typically the film coefficient and *VALUE2* (below) is typically the bulk temperature. If *Lab* = CONV and *VALUE* = *-N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* (**MP (p. 1160)**). (See the **SCOPT (p. 1649)** command for a way to override this option and use *-N* as the film coefficient.) The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements.

If **KBC (p. 970)**,0 has been issued for ramped loads, it affects only *VALUE2*, the bulk temperature (p. 1741), and the film coefficient specification is unaffected.

If *Lab* = RAD, *VALUE* is the surface emissivity.

If *Lab* = PORT, *VALUE* is a port number representing a waveguide exterior port. The port number must be an integer between 1 and 50. For acoustic 2×2 transfer admittance matrix, the port number can be any positive integer. The smaller port number corresponds to the port 1 of the 2×2 transfer admittance matrix and the greater port number corresponds to the port 2. If one port of the transfer admittance matrix is connecting to the acoustic-structural interaction interface, the port number corresponds to the port 2 of the transfer admittance matrix. A pair of ports of the 2×2 transfer admittance matrix must be defined in the same element.

If *Lab* = SHLD, *VALUE* is the surface normal velocity in harmonic analysis and the surface normal acceleration in transient analysis for acoustics.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *Lab* = IMPD, *VALUE* is resistance in (N)(s)/m^3 if *VALUE* > 0 and is conductance in mho if *VALUE* < 0 for acoustics. In acoustic transient analyses, *VALUE2* is not used.

If *Lab* = RDSF, *VALUE* is the emissivity value; the following conditions apply: If *VALUE* is between 0 and 1, apply a single value to the surface. If *VALUE* = -*N*, the emissivity may be a function of the temperature, and is determined from the EMISS property table for material *N* (**MP (p. 1160)**). The material *N* does not need to correlate with the underlying solid thermal elements.

If *Lab* = FSIN in a unidirectional Mechanical APDL to CFX analysis, *VALUE* is not used.

If *Lab* = ATTN, *VALUE* is the absorption coefficient of the surface.

VALUE2

Second surface load value (if any).

If *Lab* = CONV:

VALUE2 is typically the bulk temperature for thermal analyses.

If **KBC (p. 970)**,0 has been issued for ramped loads, the bulk temperature is ramped from the value defined by **TUNIF (p. 2027)** to the value specified by *VALUE2* for the first loadstep. If tabular boundary conditions (p. 1741) are defined, the **KBC (p. 970)** command is ignored and tabular values are used.

For acoustic analyses, VALUE2 is not used.

If *Lab* = RAD*VALUE2* is ambient temperature.

If *Lab* = SHLD, *VALUE2* is the phase angle of the normal surface velocity (defaults to zero) for harmonic response analyses while *VALUE2* is not used for transient analyses in acoustics.

If *Lab* = IMPD, *VALUE2* is reactance in (N)(s)/m^3 if *VALUE* > 0 and is the product of susceptance and angular frequency if *VALUE* < 0 for acoustics.

If *Lab* = RDSF, *VALUE2* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will also occur to ambient. If *VALUE2* is negative, radiation direction is reversed and will occur inside the element for the flagged radiation surfaces.

If *Lab* = FSIN in a unidirectional Mechanical APDL to CFX analysis, *VALUE2* is the surface interface number (not available from within the GUI).

Notes

Surface loads may be transferred from areas to elements with the **SFTRAN (p. 1782)** or **SBCTRAN (p. 1647)** commands. See the **SFGRAD (p. 1772)** command for an alternate tapered load capability.

Tabular boundary conditions

Tabular boundary conditions (*VALUE* = %*tabname*% and/or *VALUE2* = %*tabname*%) are available for the following surface load labels (*Lab*) only: PRES (real and/or imaginary components), CONV

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(film coefficient and/or bulk temperature) or HFLUX, and RAD (surface emissivity and ambient temperature). Use the ***DIM (p. 530)** command to define a table.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppImped_E>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>AppShield>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppImped_E>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>AppShield>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Areas

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Areas

Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Areas

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Areas

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Areas

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Areas

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Areas

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Areas

The SFA,,,ATTN command cannot be accessed from a menu.

SFACT, *TYPE*

Allows safety factor or margin of safety calculations to be made.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TYPE

Type of calculation:

0

No nodal safety factor or margin of safety calculations.

1

Calculate and store safety factors in place of nodal stresses.

2

Calculate and store margins of safety in place of nodal stresses.

Command Default

No nodal safety factor or margin of safety calculations.

Notes

Allows safety factor (SF) or margin of safety (MS) calculations to be made for the average nodal stresses according to:

SF = SALLOW/|Stress|

MS = (SALLOW/|Stress|) -- 1.0

Calculations are done during the display, select, or sort operation in the active coordinate system (**RSYS (p. 1639)**) with results stored in place of the nodal stresses. Use the **PRNSOL (p. 1505)** or **PLNSOL (p. 1425)** command to display the results.

Note:

The results are meaningful only for the stress (SIG1, SIGE, etc.) upon which **SALLOW (p. 1644)** is based. Nodal temperatures used are those automatically stored for the node. Related commands are **SFCALC (p. 1747)**, **SALLOW (p. 1644)**, **TALLOW (p. 1897)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>Safety Factor>Restore NodeStrs

Main Menu>General Postproc>Safety Factor>SF for Node Strs

SFADELE, *AREA*, *LKEY*, *Lab*

Deletes surface loads from areas.

SOLUTION (p. 38): Solid Surface Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Area to which surface load deletion applies. If ALL, delete load from all selected areas (**ASEL (p. 192)**). If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *AREA*.

LKEY

Load key associated with surface load (defaults to 1). See the **SFA (p. 1739)** command for details.

Lab

Valid surface load label. If ALL, use all appropriate labels. See the **SFA (p. 1739)** command for labels.

Notes

Deletes surface loads (and all corresponding finite element loads) from selected areas.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped_E>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppShield>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Areas

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Areas

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Areas

Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Areas

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped_E>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppShield>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Areas

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Areas

Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Areas

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Areas

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Areas

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Areas

Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Areas

Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Areas

Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Areas

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Areas

Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Areas

---

SFALIST, *AREA*, *Lab*

Lists the surface loads for the specified area.

SOLUTION (p. 38): Solid Surface Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

AREA

Area at which surface load is to be listed. If ALL (or blank), list for all selected areas (**ASEL (p. 192)**). If *AREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *AREA*.

Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. See the **SFA (p. 1739)** command for labels.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Areas

Utility Menu>List>Loads>Surface Loads>On Picked Areas

SFBEAM, *Elem*, *LKEY*, *Lab*, *VALI*, *VALJ*, *VAL2I*, *VAL2J*, *IOFFST*, *JOFFST*, *LENRAT*

Specifies surface loads on beam and pipe elements.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Elem

Element to which surface load is applied. If ALL, apply load to all selected beam elements (**ESEL (p. 661)**). If *Elem* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted in *Elem*.

LKEY

Load key associated with surface load (defaults to 1). Load keys (1, 2, 3, etc.) are listed under "Surface Loads" in the input table for each element type in the *Element Reference*. For beam and some pipe elements, the load key defines the load orientation.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*. Structural labels: PRES (pressure).

VALI, VALJ

Surface load values at nodes I and J. If *VALJ* is blank, it defaults to *VALI*. If *VALJ* is zero, a zero is used.

VAL2I, VAL2J

Second surface load values at nodes I and J. Currently not used.

IOFFST, JOFFST

Offset distance from node I (toward node J) where *VALI* is applied, and offset distance from node J (toward node I) where *VALJ* is applied, respectively.

LENRAT

Offset distance flag:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Offset is in terms of length units (default).

1

Offset is in terms of a length ratio (0.0 to 1.0).

Notes

Specifies surface loads on the selected beam elements. Distributed loads are applied on a force-per-length basis (that is, the width of the underlying element is not considered). To list and delete surface loads applied with this command, use the **SFELIST (p. 1770)** and **SFEDELE (p. 1768)** commands, respectively.

If no offset values (*IOFFSET* and *JOFFSET*) are specified, the load is applied over the full element length. Values may also be input as length fractions, depending on the *LENRAT* setting. For example, assuming a line length of 5.0, an *IOFFST* of 2.0 with *LENRAT* = 0 or an *IOFFST* of 0.4 with *LENRAT* = 1 represent the same point. If *JOFFST* = -1, *VALI* is assumed to be a point load at the location specified via *IOFFST*, and *VALJ* is ignored. (*IOFFSET* cannot be equal to -1.) The offset values are stepped even if you issue a **KBC (p. 970)**,0 command.

Offsets are only available for element types BEAM188 and PIPE288 if using the cubic shape function (KEYOPT(3) = 3) for those element types.

To accumulate (add) surface loads applied with this command, use the **SFCUM (p. 1758)**,,ADD command. Use the same offset values used on the previous **SFBEAM (p. 1746)** command (for a given element face); otherwise, the loads do not accumulate. If no offsets are specified, the command applies the previous offset values.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Beams

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Beams

---

SFCALC, *LabR*, *LabS*, *LabT*, *TYPE*

Calculates the safety factor or margin of safety.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LabS

Labeled result item corresponding to the element stress.

LabT

Labeled result item corresponding to the element temperature.

TYPE

Type of calculation:

0 or 1

Use safety factor (SF) calculation.

2

Use margin of safety (MS) calculation.

3

Use 1/SF calculation.

Notes

Calculates safety factor (SF) or margin of safety (MS) as described for the **SFACT (p. 1743)** command for any labeled result item (see **ETABLE (p. 687)** command) for the selected elements. Use the **PRETAB (p. 1490)** or **PLETAB (p. 1411)** command to display results. Allowable element stress is determined from the SALLOW-TALLOW table (**SALLOW (p. 1644)**, **TALLOW (p. 1897)**).

Menu Paths

Main Menu>General Postproc>Safety Factor>SF for ElemTable

SFCONTROL, *KCSYS*, *LCOMP*, *VAL1*, *VAL2*, *VAL3*, *KTAPER*, *KUSE*, *KAREA*, *KPROJ*, *KFOLLOW*

Defines structural surface-load properties on selected elements and nodes for subsequent loading commands.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCSYS

Specifies how the load direction is determined:

0 (or blank) – Use the coordinate system of an element face. A local coordinate system is projected onto the face, if defined (*VAL1*) (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 – Use a local coordinate system. A local coordinate system must be defined and is not projected onto the face.

2 – Use a custom (user-defined) vector in the global Cartesian coordinate system.

LCOMP

Load-component definition when *KCSYS* = 0 or 1. The following table shows how the component (or primary direction) is determined based on the coordinate system (p. 1751):

LCOMP [a]	**Direction**		
	KCSYS = 0		*KCSYS* = 1[b]
	VAL1 = 0	*VAL1* > 10[c]	--
0	Normal	Normal	x direction
1	x direction[d]	x direction[e]	y direction
2	y direction[d]	y direction[e]	z direction

[a] Because *KCSYS* = 0 and *KCSYS* = 1 use the same *LCOMP*, the most recent load-component definition overwrites the previous component even if *KCSYS* differs.

[b] The local coordinate system is not projected onto the element face.

[c] *VAL1* < 0 is not valid. 0 < *VAL1* < 11 is ignored.

[d] The positive x and y directions are determined by the default surface coordinate directions. (See Figure 18: Coordinate System for Load Application on the Faces of 3D Solid and Shell Elements (p. 1751).)

[e] The positive x and y directions are determined by the user-defined local coordinate system.

VAL1, VAL2, VAL3

When *KCSYS* = 0:

VAL1 – Determines the first tangential axis (x axis). Not valid for the edges of 3D shell or 2D elements.

0 – Aligns the x axis to the first parametric direction $\overline{\xi}$ (default).

1 – Aligns x axis to the second parametric direction $\overline{\eta}$.

>10 – ID of the local coordinate system. The local coordinate system is projected to selected face of the element.

VAL2 – Not used.

VAL3 – Rotation angle of a tangential load (optional). Not valid for the edges of 3D shell or 2D elements. If this value is specified, the tangential load rotates further with respect to the surface normal. The load component (*LCOMP*) becomes the reference axis to rotate.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When *KCSYS* = 1:

VAL1 – ID of the local coordinate system for the load. The axes of the local coordinate system are fixed in the global cartesian coordinate system.

VAL2 – *Not used.*

VAL3 – *Not used.*

When *KCSYS* = 2:

VAL1, *VAL2*, *VAL3* – The X / Y / Z components, respectively, of the direction vector in the global Cartesian coordinate system.

KTAPER

Global tapered load behavior (valid for **SFE (p. 1761)** only):

0 – Load does not vary (default).

1 – Load varies with respect to the current element locations. The magnitude changes with respect to the element deformation.

2 – Load varies with respect to the initial element locations. The load magnitude for each element remains constant throughout the solution. .

For more information, see "Global Tapered Load Behavior (*KTAPER*)" (p. 1755).

KUSE

Load direction with respect to the surface normal of the selected face:

0 – Use the load as calculated (default).

1 – Use a positive load only (negative set to zero, valid for *LCOMP* = 0 and *KCSYS* = 0).

2 – Use a negative load only (positive set to zero, valid for *LCOMP* = 0 and *KCSYS* = 0).

3 – Applied load is not used if the surface normal is oriented in the same general direction as the user-defined vector. Valid for *KCSYS* = 2 only.

KAREA

Loaded area during large deformation:

0 – Use the current (deformed) area (default).

1 – Use the initial area.

KPROJ

Vector-oriented load (*KCSYS* = 2) behavior:

0 – Apply the load on the full area and include the tangential component (default).

1 – Apply the load on the projected area and include the tangential component.

2 – Apply the load on the projected area and exclude the tangential component.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KFOLLOW

Controls follower-load behavior. Valid when *KCSYS* = 1 or 2, or when *KCSYS* = 0 and *VAL1* > 10.

0 – The load maintains a fixed direction (default).

1 –The load follows the element deformation.

For more information, see "Follower Load Behavior" (p. 1755).

Notes

SFCONTROL (p. 1748) defines the properties of structural distributed loads for all subsequent **SF (p. 1733)** or **SFE (p. 1761)** loading commands. (**SFA (p. 1739)** and **SFL (p. 1774)** are not supported.)

The command does not support 2D-to-3D analysis.

To update a load property or properties, reissue **SFCONTROL (p. 1748)** with the new option(s) before issuing further **SF (p. 1733)** or **SFE (p. 1761)** commands.

SFCONTROL (p. 1748) supports these structural solid, shell, solid-shell, and coupled-field elements:

SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, SHELL281, SOLID285, PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227.

To list the current set of control data, issue **SFCONTROL (p. 1748)**,STAT.

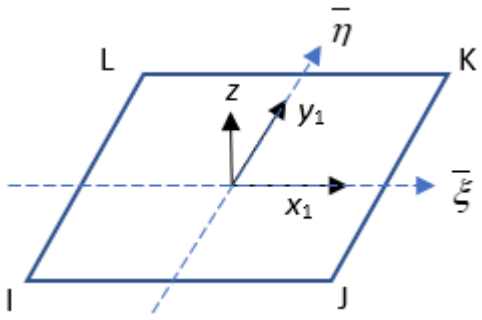
To reset all input values to defaults, issue **SFCONTROL (p. 1748)**,NONE.

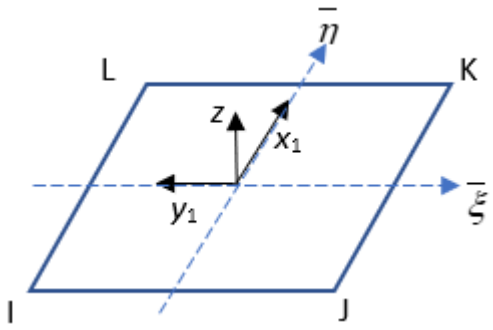
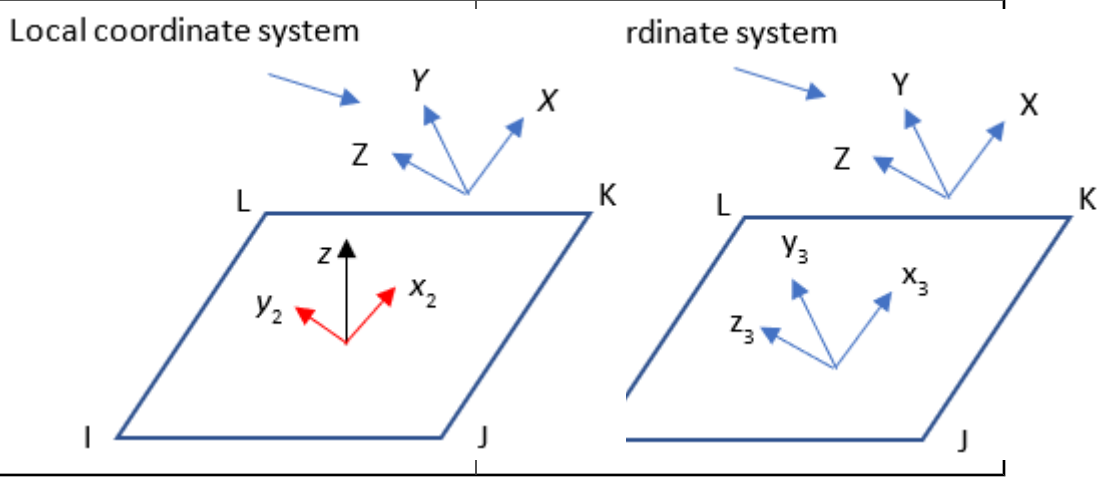
When *KCSYS* = 0, the positive normal load acts in the negative surface normal (-*z*) direction. The positive tangential loads act in the positive coordinate direction. A user-defined coordinate system (*VAL1*) is ignored if the load is applied on an edge of a plane element (PLANE182, PLANE183, SHELL208, SHELL209, PLANE222, and PLANE223).

When *KCSYS* = 1, the loading direction follows the positive direction of the local coordinate system. The ID of a local coordinate system (*VAL1*) is required.

The following figure shows how the coordinate directions are determined on a face of a solid or shell element with different *KCSYS* and *VAL1* input:

Figure 18: Coordinate System for Load Application on the Faces of 3D Solid and Shell Elements

	KCSYS = 0	*KCSYS* = 1
VAL1 = 0		–

VAL1 = 1		–
VAL1 > 10		

where:

z, x_1, y_1 : normal and tangential directions (default surface coordinate system)

z, x_2, y_2 : normal and user-defined tangential directions (user-defined surface coordinate system)

x_3, y_3, z_3 : x, y, and z directions defined by the local coordinate system when *KCSYS* = 1

When *KCSYS* = 0:

The parametric direction determines the default coordinate system on a face. By default, the first parametric direction ($\overline{\xi}$) becomes the first tangential direction (x direction). If *VAL1* = 1, the second parametric direction ($\overline{\eta}$) is selected as the x direction.

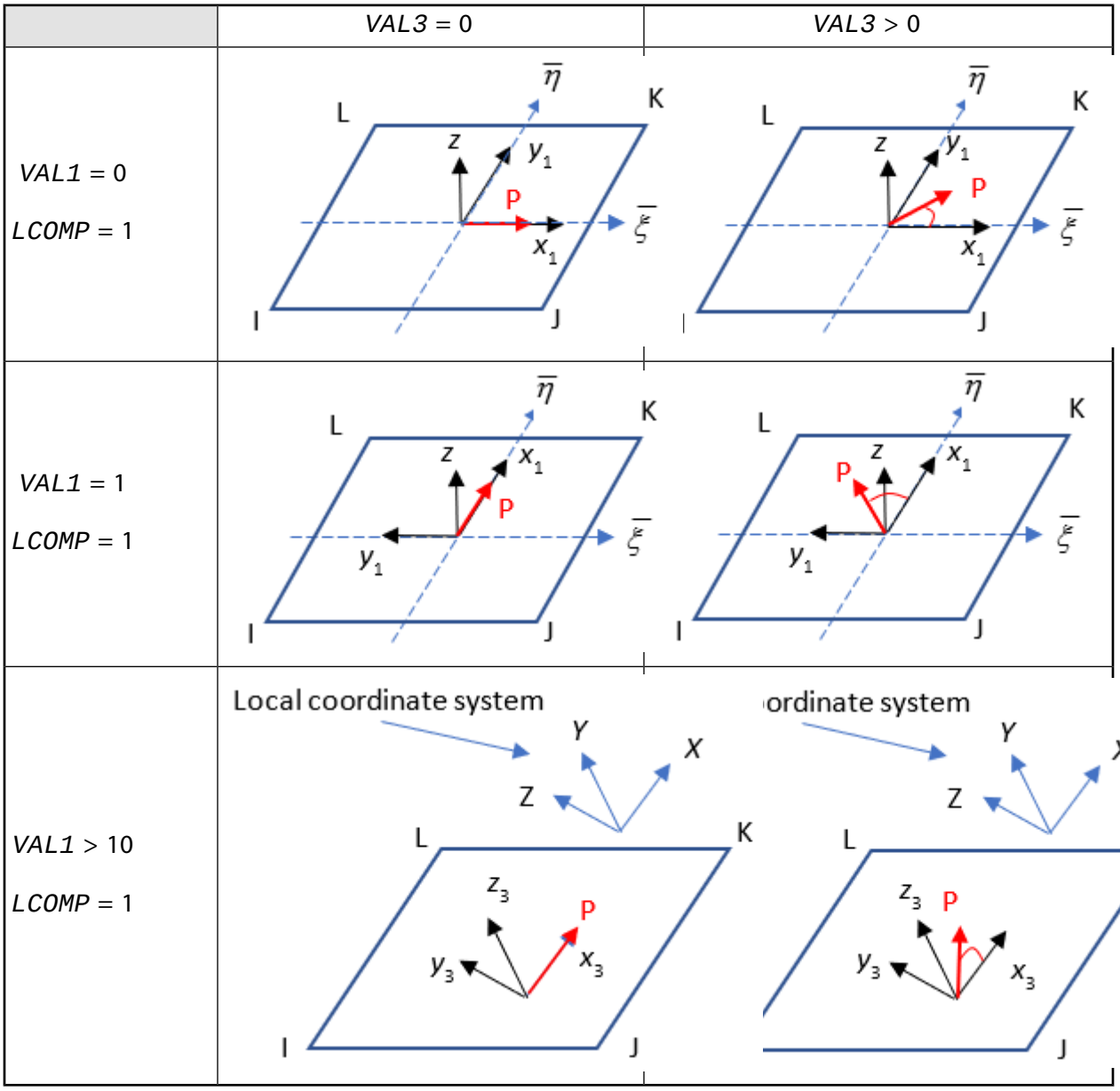
For the projected local coordinate system (*VAL1* > 10), the direction of the first tangential axes (x_2) is determined by the projection of the local coordinate system onto the face. The projected tangential axes may rotate if the direction of the face normal (z) changes in the space during solution. If the direction of the face normal (z) is fixed (for in-plane rotation, for example), the tangential axes do not follow element deformation. To enable your coordinate system to always follow the element deformation, specify the follower option (*KFOLLOW*).

Coordinate System vs. Load Direction

For the loads defined in the default coordinate system (*KCSYS* = 0), the tangential direction of a load is restricted because it is aligned to the direction of the axis (*LCOMP*). The load direction can be arbitrary by adding additional rotation to the load (*VAL3*). The following figure shows how the load direction is determined on the face of an element. If *VAL3* > 0, *LCOMP* becomes the reference axis to define the rotation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19: Load Direction in the Default Coordinate System



When *KCSYS* = 1:

The positive direction of the x_3, y_3, and z_3 axes follow the direction of the local coordinate system without adjustment. Therefore, they do not follow element deformation.

When *KCSYS* = 2:

The loading direction follows the positive direction of the orientation vector, defined via *VAL1*, *VAL2*, and *VAL3*. The direction is calculated as $VAL1\bar{i}+VAL2\bar{j}+VAL3\bar{k}\Big/\sqrt{VAL1^2+VAL2^2+VAL3^2}$, where $\bar{i}$, $\bar{j}$, and $\bar{k}$ are unit vectors in the global Cartesian coordinate system. The loading direction is fixed and does not follow the deformation of a face of a selected element.

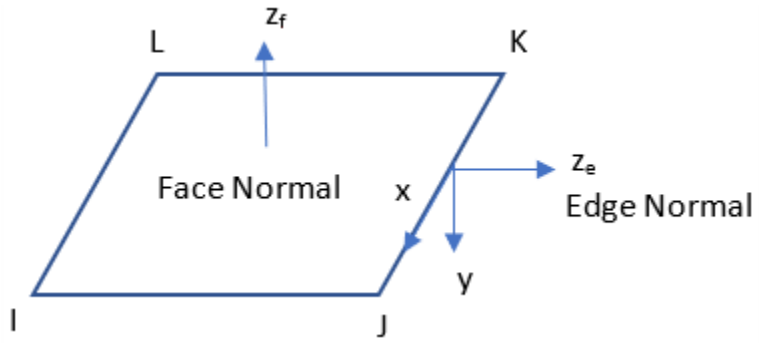
You can adjust the load magnitude (*KUSE* and *KPROJ*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the follower option (*KFOLLOW*) to specify whether the load has a fixed direction (*KCSYS* = 1 or 2), or projects the local coordinate system to the element face to follow element deformation (*KCSYS* = 0 and *VAL1* > 10) .

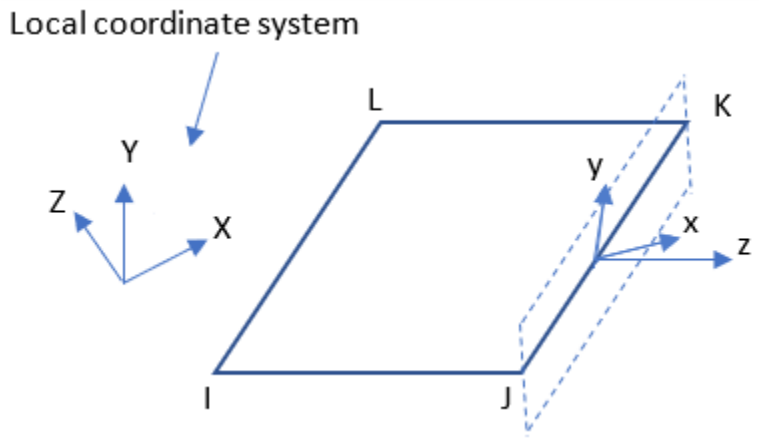
The following figure shows the default direction of the normal (z) and tangential (x) components on an edge of a 3D shell element (SHELL181 or SHELL281).

Figure 20: Default Coordinate System of Surface Load on the Edge of a 3D Shell Element



If a local coordinate system is defined and *KCSYS* = 0 for the edge of the 3D shell element, the tangential directions are adjusted in the plane of the edge.

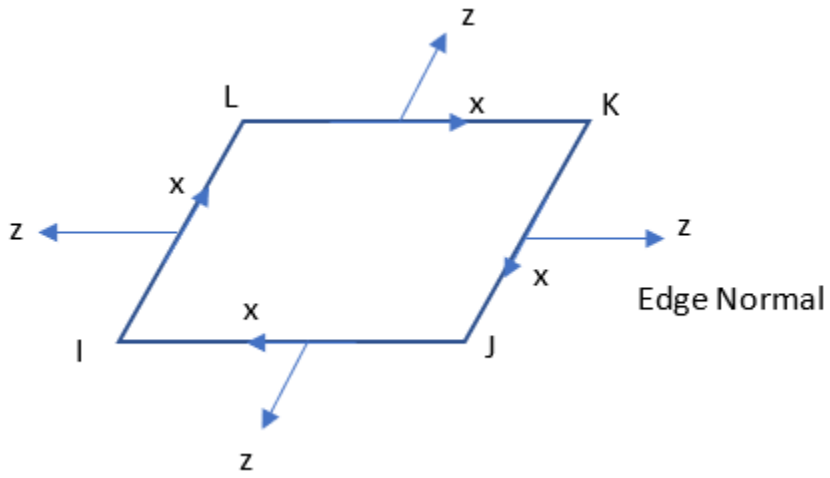
Figure 21: Projected Coordinate System of Surface Load on the Edge of a 3D Shell Element



The following figure shows the positive direction of the loads on the edge of a plane element when *KCSYS* = 0. The positive direction of the tangential load is defined by the definition of the faces (J-I, K-J, L-K, I-L).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22: Positive Tangential Load on the Edge of a Plane Element



If *KCSYS* = 1 or 2, the load direction does not change and *KUSE* = 1 or 2 is ignored.

If *KCSYS* = 1 or 2, or *KTAPER* > 0, you cannot specify a gradient (slope) for surface loads (**SFGRAD (p. 1772)**) or a varying surface load (**SFFUN (p. 1771)**).

This command is also valid in the PREP7 processor.

Global Tapered Load Behavior (*KTAPER*)

If *KTAPER* = 1, the magnitude of the load is determined by the *current* location of a point:

$$P=P_I+P_JX+P_KY+P_LZ$$

where:

$P_I \sim P_L$: Values on **SFE (p. 1761)** (*VAL1* ~ *VAL4*)

X, Y, Z : Global Cartesian coordinates at the current location of the point.

If *KTAPER* = 2, the magnitude of the load is determined by the *initial* location of a point:

$$P=P_I+P_JX_0+P_KY_0+P_LZ_0$$

where:

$P_I \sim P_L$: Values on **SFE (p. 1761)** (*VAL1* ~ *VAL4*)

X_0, Y_0, Z_0 : Global Cartesian coordinates at the initial location of the point.

KTAPER is not valid for use with **SF (p. 1733)**.

Follower Load Behavior

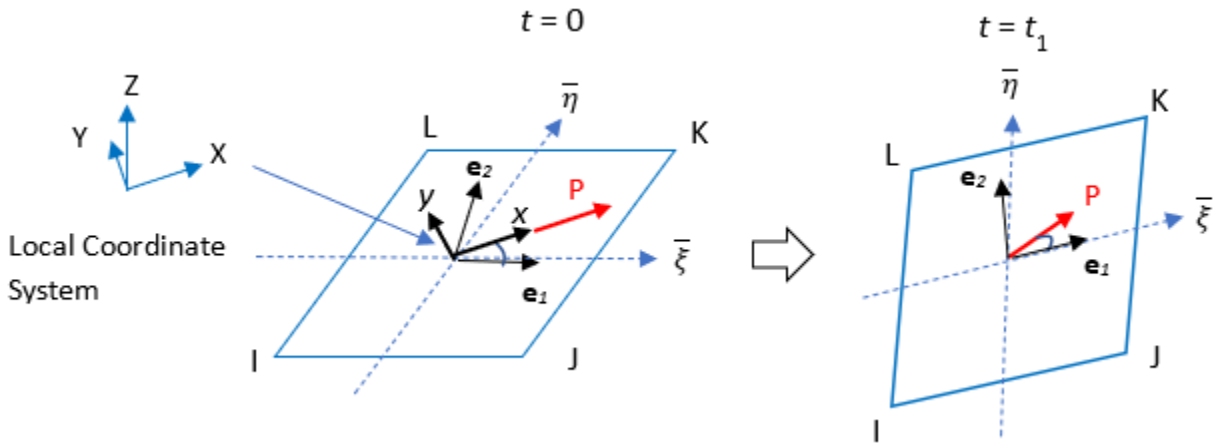
The follower option (*KFOLLOW*) determines whether the load maintains a fixed direction (default) or follows element deformation. The option applies to surface loads defined by a fixed direction (*KCSYS* = 1 or 2) or by a projected user-defined coordinate system (*KCSYS* = 0 and *VAL1* > 10).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When *KCSYS* = 0 and *VAL1* > 10:

The selected local coordinate system is projected onto the face at the initial state, then the projected tangential component (*LCOMP*) is attached to the orthonormal basis ($\mathbf{e}_1$, $\mathbf{e}_2$, $\mathbf{e}_3$) of the face. The orthonormal basis may or may not be coincident to the $\bar{\xi}-\bar{\eta}$ coordinate system. During solution, the basis is updated at the current time step and the load direction is updated with respect to the basis.

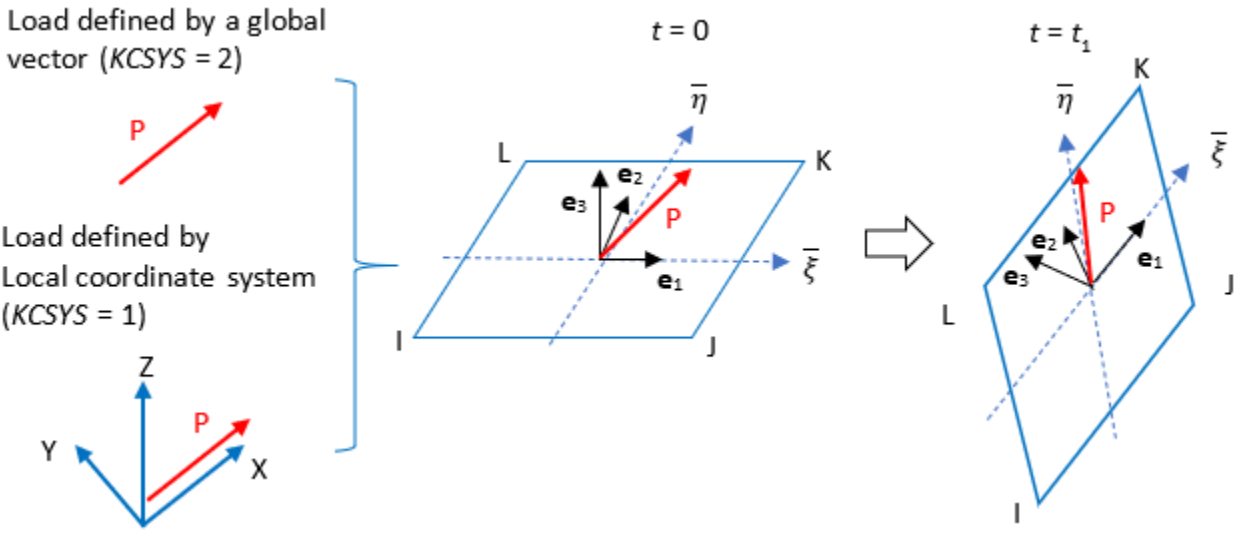
Figure 23: Follower Load Behavior in the Projected Coordinate System



When *KCSYS* = 1 or 2:

The global direction vector is attached to the orthonormal basis of selected face at the initial state. When *KCSYS* = 1, the direction of selected axis is considered as the direction vector and attached to the basis. During solution, the basis is updated at current time step and the load direction is updated with respect to the basis.

Figure 24: Follower Load Behavior for a User-Defined Orientation

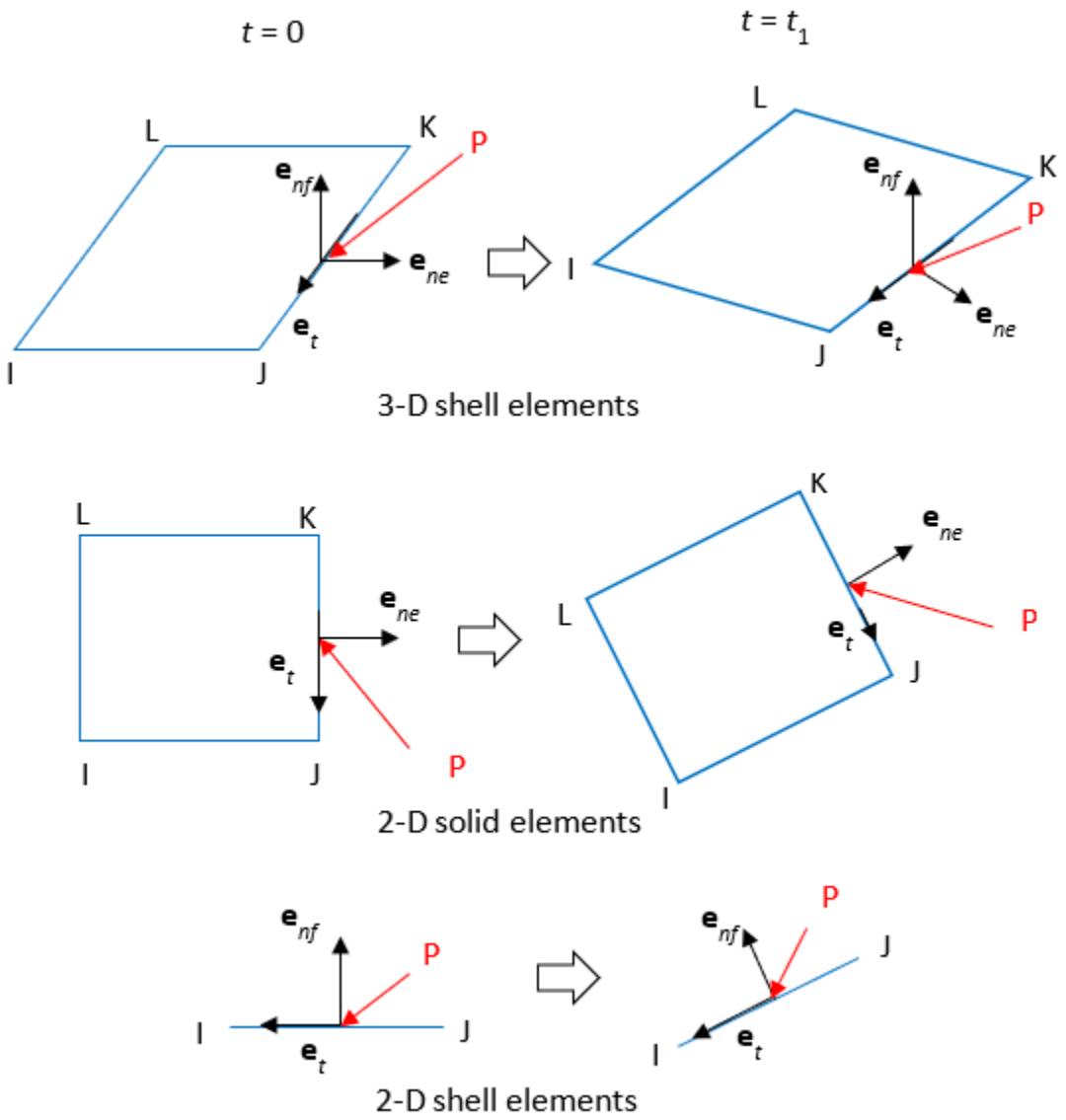


Follower loads on the edges of 3D shell and 2D elements:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The load is attached to the basis defined on the edge ($\mathbf{e}_t$, $\mathbf{e}_{ne}$, $\mathbf{e}_{nf}$ [tangential, edge-normal, and face-normal vector, respectively]). For 2D solid elements, the face-normal orientation ($\mathbf{e}_{nf}$) is that of the global Z axis. For 2D shell elements, the edge-normal ($\mathbf{e}_{ne}$) is that of the global Z axis.

Figure 25: Follower Load Behavior on the Edges of 3D Shell and 2D Elements



If it is necessary to use the follower option after the first load step or at the restart analysis, define the load direction with respect to the current geometry (that is, the current basis of an element face). The follower option for the local coordinate system (*KCSYS* = 1) is not allowed after the first load step or at the restart analysis.

Load-Stiffness Effects

Load-stiffness effects are included in the supported elements for the real part of all loads at the current configuration. All other load properties are included in the load vector of the elements.

You can specify an unsymmetric matrix (**NROPT (p. 1261)**,UNSYM) for the load-stiffness effects if needed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

SFCUM, *Lab*, *Oper*, *FACT*, *FACT2*

Specifies that surface loads are to be accumulated.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid surface load label. If ALL, use all appropriate labels.

Discipline	Surface Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
Substructure	SELV	load vector number
Electric	CHRGS	surface charge density
	MXWF	Maxwell force flag
Magnetic	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for INFIN110 and INFIN111
Diffusion	DFLUX	Diffusion flux

1. Thermal labels CONV and HFLUX are mutually exclusive.

Oper

Accumulation key:

REPL

Subsequent values replace the previous values (default).

ADD

Subsequent values are added to the previous values.

IGNO

Subsequent values are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FACT

Scale factor for the first surface load value. A (blank) or '0' entry defaults to 1.0.

FACT2

Scale factor for the second surface load value. A (blank) or '0' entry defaults to 1.0.

Command Default

Replace previous values.

Notes

Allows repeated surface loads (pressure, convection, etc.) to be replaced, added, or ignored. Surface loads are applied with the **SF (p. 1733)**, **SFE (p. 1761)**, and **SFBEAM (p. 1746)** commands. Issue the **SFELIST (p. 1770)** command to list the surface loads. The operations occur when the next surface load specifications are defined. For example, issuing the **SF (p. 1733)** command with a pressure value of 25 after a previous **SF (p. 1733)** command with a pressure value of 20 causes the current value of that pressure to be 45 with the add operation, 25 with the replace operation, or 20 with the ignore operation. All new pressures applied with **SF (p. 1733)** after the ignore operation will be ignored, even if no current pressure exists on that surface.

Scale factors are also available to multiply the next value before the add or replace operation. A scale factor of 2.0 with the previous "add" example results in a pressure of 70. Scale factors are applied even if no previous values exist. Issue **SFCUM (p. 1758)**,STAT to show the current label, operation, and scale factors. Solid model boundary conditions are not affected by this command, but boundary conditions on the FE model are affected.

Note:

The FE boundary conditions may still be overwritten by existing solid model boundary conditions if a subsequent boundary condition transfer occurs.

SFCUM (p. 1758) does not work for tabular boundary conditions.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Replace vs Add>Surface Loads

Main Menu>Solution>Define Loads>Settings>Replace vs Add>Surface Loads

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SFDELE,*Nlist*,*Lab*

Deletes surface loads.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Nlist

Label defining where to find the list of nodes:

ALL

Use all selected nodes (**NSEL (p. 1266)**). If P, use graphical picking in GUI. A component label may be substituted for *Nlist*.

Lab

Valid surface load label. If ALL, use all appropriate labels. See the **SF (p. 1733)** command for labels.

Notes

Deletes surface loads as applied via **SF (p. 1733)**. Loads are deleted only for the specified nodes on external faces of selected area and volume elements. For shell elements, if the specified nodes include face one (which is usually the bottom face) along with other faces (such as edges), only the loads on face one will be deleted. The element faces are determined from the list of selected nodes as described for **SF (p. 1733)**. Issue **SFEDELE (p. 1768)** to delete loads explicitly by element faces.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped_E>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppShield>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Node Components

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Nodes

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped_E>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppShield>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Nodes

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Nodes

Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Nodes

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Nodes

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Nodes

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Nodes

Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Node Components

Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Nodes

Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Nodes

Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Nodes

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Nodes

Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Nodes

---

SFE, *Elem*, *LKEY*, *Lab*, *KVAL*, *VALUE1*, *VALUE2*, *VALUE3*, *VALUE4*, *MESHFLAG*

Defines surface loads on elements.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Elem

Element to which surface load applies. If ALL, apply load to all selected elements (**ESEL (p. 661)**). If *Elem* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *Elem*.

LKEY

Load key or face number associated with surface load (defaults to 1). Load keys (1,2,3, etc.) are listed under "Surface Loads" in the input data table for each element type in the *Element Reference*.

If you issue **SFCONTROL (p. 1748)** before **SFE (p. 1761)**, *LKEY* is the face number for supported structural solid and shell elements (p. 1751).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*.

Discipline	Surface Load Label	Label Description
Structural	PRES	Pressure
	FREQ	Frequency (harmonic analyses only)
Thermal	CONV[1]	Convection
	HFLUX[1]	Heat flux
	RAD	Radiation
	RDSF	Surface-to-surface radiation
Fluid	FSI[2]	Fluid-structure interaction flag
	IMPD	Impedance boundary flag
Electric	CHRGS	Surface charge density
	MXWF	Maxwell force flag
Magnetic	MXWF	Maxwell force flag
Acoustic fluid	FSI[2]	Fluid-structure interaction flag
	IMPD	Impedance or admittance coefficient
	SHLD	Surface normal velocity or acceleration
	MXWF	Maxwell surface flag or equivalent source surface
	FREE	Free surface flag
	INF	Exterior Robin radiation boundary flag
	PORT	Port number
	ATTN	Absorption coefficient or transmission loss
	BLI	Viscous-thermal boundary layer surface flag
Field-surface interface	FSIN	Field-surface interface number
Poromechanics	FFLX	Fluid flow flux
Infinite element	INF	Exterior surface flag for INFIN110 and INFIN111
Substructure	SELV[3]	Load vector number
Diffusion	DFLUX	Diffusion flux

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID29, FLUID30, FLUID220, and FLUID221 elements.
3. When a load vector exists for a thermal superelement, it must be applied and have a scale factor of 1 (**SFE (p. 1761)**,,,,SELV,,1).

KVAL

Value key. If *Lab* = PRES:

0 or 1

VALUE1 through *VALUE4* are used as real components of pressures.

2

VALUE1 through *VALUE4* are used as imaginary components of pressures.

Value key. If *Lab* = CONV:

0 or 1

For thermal analyses, *VALUE1* through *VALUE4* are used as the film coefficients.

2

For thermal analyses, *VALUE1* through *VALUE4* are the bulk temperatures.

3

VALUE1 through *VALUE4* are used as film effectiveness.

4

VALUE1 through *VALUE4* are used as free stream temperature.

Value key. If *Lab* = RAD:

0 or 1

VALUE1 through *VALUE4* are used as the emissivities.

2

VALUE1 through *VALUE4* are ambient temperatures.

Value key. If *Lab* = RDSF:

0 or 1

VALUE1 is the emissivity value between 0 and 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2

VALUE1 is the enclosure number.

Value key. If *Lab* = IMPD:

0 or 1

For acoustic harmonic analyses, VALUE1 through VALUE4 are used as the real part of the impedance.

2

For acoustic harmonic analyses, VALUE1 through VALUE4 are used as the imaginary part of the impedance.

Value key. If *Lab* = SHLD:

0 or 1

For acoustic analyses, *VALUE1* through *VALUE4* are used as the normal velocity (harmonic) or normal acceleration (transient).

2

For acoustic analyses, *VALUE1* through *VALUE4* are used as the phase angle for harmonic response analyses.

Value key. If *Lab* = ATTN:

0 or 1

For acoustic analyses, *VALUE1* through *VALUE4* are used as the absorption coefficient of the surface.

2

For acoustic analyses, *VALUE1* through *VALUE4* are used as the transmission loss (dB) of the coupled wall in an energy diffusion solution for room acoustics.

Value key. If *Lab* = SELV:

0 or 1

VALUE1 is the multiplier on real load vector *LKEY*.

2

VALUE1 is the multiplier on imaginary load vector *LKEY*.

If only one set of data is supplied, the other set of data defaults to previously specified values (or zero if not previously specified) in the all of the following cases:

- Emissivities are supplied and *Lab* = RAD

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Temperatures are supplied and *Lab* = RAD
- Temperatures are supplied and *Lab* = CONV
- Film coefficients are supplied and *Lab* = CONV
- Normal velocity/acceleration for acoustics is supplied and *Lab* = SHLD
- Phase angle for acoustics is supplied and *Lab* = SHLD

Not applicable for *Lab* = FREQ, CHRGS, PORT, FSIN, FFLX, DFLUX

VALUE1

First surface load value (typically at the first node of the face), or the name of a table for specifying tabular boundary conditions.

Face nodes are listed in the order given for **Surface Loads** in the input data table for each element type in the *Element Reference*. For example, for SOLID185, the item 1-JILK associates *LKEY* = 1 (face 1) with nodes J, I, L, and K. Surface load value *VALUE1* then applies to node J of face 1. To specify a table, enclose the table name in percent signs (%), for example, %*tabname*%. Use the ***DIM (p. 530)** command to define a table. Only one table can be specified, and it must be specified in the *VALUE1* position; tables specified in the *VALUE2*, *VALUE3*, or *VALUE4* positions are ignored. *VALUE2* applies to node I, etc.

If *Lab* = PRES and *KVAL* = 2, this value is the imaginary pressure component, used by the following supported elements:

- Surface elements: SURF153, SURF154 and SURF159.
- Solid and solid-shell elements: PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, and SOLID285.
- Shell elements: SHELL181, SHELL281, SHELL208, and SHELL209.
- Coupled-field elements with structural degrees of freedom: PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227.

If *Lab* = CONV, *KVAL* = 0 or 1, and *VALUE1* = -*N*, the film coefficient is assumed to be a function of temperature and is determined from the HF property table for material *N* (**MP (p. 1160)**). (See the **SCOPT (p. 1649)** command for a way to override this option and use -*N* as the film coefficient.) The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements.

If *Lab* = CONV, *KVAL* = 2, *VALUE1* specifies the bulk temperature. If **KBC (p. 970)**,0 has been issued for ramped loads, the bulk temperature is ramped from the value defined by **TUNIF (p. 2027)** to that specified on *VALUE1* (for the first loadstep). If a table name is specified for *VALUE1*, the **KBC (p. 970)** command is ignored and tabular values are used.

If *Lab* = PORT, *VALUE1* is a port number representing a waveguide port. The port number must be an integer between 1 and 50. For an acoustic 2×2 transfer admittance matrix, the port number can be any positive integer. The smaller port number corresponds to port 1 of the 2×2 transfer admittance matrix, and the greater port number corresponds to port 2. If one port of the transfer admittance matrix is connecting to the acoustic-structural interaction interface, the port number cor-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

responds to port 2 of the transfer admittance matrix. A pair of ports of the 2×2 transfer admittance matrix must be defined in the same element.

If *Lab* = RDSF, *KVAL* = 0 or 1, and *VALUE1* = -*N*, the emissivity is assumed to be a function of the temperature, and is determined from the EMISS property table for material *N* (**MP (p. 1160)**). The material *N* does not need to correlate with the underlying solid thermal elements. If *Lab* = RDSF, *KVAL* = 2, and *VALUE1* is negative, radiation direction is reversed and will occur inside the element for the flagged radiation surfaces.

If *Lab* = FSIN in a unidirectional Mechanical APDL-to-CFX analysis, *VALUE1* is not used.

If *Lab* = SELV, *VALUE1* represents the scale factor (default = 0.0).

If *Lab* = ATTN, *VALUE1* is the absorption coefficient.

VALUE2, VALUE3, VALUE4

Surface load values at the second, third, and fourth nodes (if any) of the face.

If all three values are blank, all default to *VALUE1*, giving a constant load. Zero or other blank values are used as zero.

If *VALUE2*, *VALUE3*, or *VALUE4* are magnitudes of the load, they are ignored if *VALUE1* is a table. If *VALUE2*, *VALUE3*, or *VALUE4* are any other values, they are used even if *VALUE1* is a table (for example, the load direction for face 5 of SURF154).

If *Lab* = FSIN in a unidirectional Mechanical APDL-to-CFX analysis, *VALUE2* is the surface interface number (not available in the GUI). *VALUE3* and *VALUE4* are not used.

MESHFLAG

Specifies how to apply normal pressure loading on the mesh. Valid in a nonlinear adaptivity analysis when *Lab* = PRES and *KVAL* = 0 or 1.

0 – Pressure loading occurs on the current mesh (default).

1 – Pressure loading occurs on the initial mesh for nonlinear adaptivity.

Notes

SFE (p. 1761) defines surface loads on selected elements.

Caution:

You cannot use **SFE (p. 1761)** with the INFIN110 or INFIN111 elements without prior knowledge of element-face orientation (that is, you must know which face is the exterior in order to flag it). Also, for surface-effect elements SURF153 and SURF154, use *LKEY* to enable tangential and other loads. For supported structural solid and shell elements (p. 1751), issue **SFCONTROL (p. 1748)** to define tangential and other loads.

SFE (p. 1761) can apply tapered loads over the faces of most elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use these related surface-load commands with **SFE (p. 1761)**:

SF (p. 1733) – Defines surface loads on *nodes*.

SFBEAM (p. 1746) – For beam elements allowing lateral surface loads that can be offset from the nodes, this command specifies the loads and offsets.

SFCONTROL (p. 1748) – Applies general (normal, tangential, and other) surface loads to supported structural elements (p. 1751).

SFCUM (p. 1758) – Accumulates (adds) surface loads applied via **SFE (p. 1761)**.

SFDELE (p. 1760) – Delete loads applied via **SFE (p. 1761)**.

SFFUN (p. 1771) – Applies loads from a node-vs.-value function.

SFGRAD (p. 1772) – Applies an alternate tapered load.

The **SFE (p. 1761)** command can also define fluid-pressure-penetration loads (*Lab* = PRES) at a contact interface. For this type of load, *LKEY* = 1 is used to specify the normal pressure values, *LKEY* = 3 is used to specify the tangential pressure values along the x direction of **ESYS (p. 685)**, *LKEY* = 4 is used to specify the tangential pressure values along the y direction of **ESYS (p. 685)**, and *LKEY* = 2 is used to specify starting points and penetrating points. See Applying Fluid-Penetration Pressure in the *Contact Technology Guide* for details on how to apply this type of load.

Film effectiveness and free-stream temperatures specified via *Lab* = CONV are valid only for SURF151 and SURF152. Film effectiveness must be between 0 and 1 and it defaults to 0. If film effectiveness is applied, bulk temperature is ignored. When film effectiveness and free stream temperatures are specified, the commands to specify a surface-load gradient (**SFGRAD (p. 1772)**) or surface-load accumulation (**SFCUM (p. 1758)**) are not valid. For more information about film effectiveness, see Conduction, Convection, and Mass Transport (Advection) in the *Theory Reference*.

You can specify a table name only when using structural (PRES) and thermal (CONV [film coefficient, bulk temperature, film effectiveness, and free stream temperature], HFLUX), diffusion flux (DFLUX), surface emissivity and ambient temperature (RAD), impedance (IMPD), normal velocity or acceleration (SHLD), absorption coefficient (ATTN), and substructure (SELV) surface load labels.

When a tabular function load is applied to an element, the load will not vary according to the positioning of the element in space.

For cases where Lab=FSI, MXWF, FREE, and INF, VALUE is not needed.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in the PREP7 and **/MAP (p. 1107)** processors.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Load Vector>For Superelement

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Element Components

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Elements>Tapered

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Elements>Uniform

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Elements

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Elements

Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Elements

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Elements

Main Menu>Solution>Define Loads>Apply>Load Vector>For Superelement

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Element Components

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Elements

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Elements>Tapered

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Elements>Uniform

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Elements

Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Elements

The SFE,,,ATTN command cannot be accessed from a menu.

SFEDELE, *ELEM*, *LKEY*, *Lab*, *Lcomp*

Deletes surface loads from elements.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element to which surface load deletion applies. If ALL, delete load from all selected elements (**ESEL (p. 661)**). If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

LKEY

Load key associated with surface load (defaults to 1). If ALL, delete surface loads for all load keys. If **SFCONTROL (p. 1748)** is issued for selected elements, this value is the face number. If ALL, deletes the surface load based on *Lcomp*.

Lab

Surface load label. If ALL, use all appropriate labels. See **SFE (p. 1761)** for valid labels.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lcomp

Label of surface-load components. Valid when the surface load is defined via **SFCONTROL (p. 1748)**. Valid labels are NORM, TANX, TAXY when *KCSYS* = 0, LOCX, LOCY, LOCZ when *KCSYS* = 1, and USER when *KCSYS* = 2. (*KCSYS* is specified when issuing **SFCONTROL (p. 1748)**.) If ALL, deletes all component on the face defined by *LKEY*.

Notes

Deletes surface loads from selected elements. See the **SFDELE (p. 1760)** command for an alternate surface load deletion capability based upon selected nodes.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Elems

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Boundary>AppImped_E>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Load Vector>For Superelement

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Element Components

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Elements

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Elements

Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Elems

Main Menu>Solution>Define Loads>Delete>Electric>Boundary>AppImped_E>On Elements

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Elements

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Elements

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Elements

Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Elements

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Elements

Main Menu>Solution>Define Loads>Delete>Load Vector>For Superelement

Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Elements

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Elements

Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Element Components

Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Elements

Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Elements

Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Elements

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Elements

Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Elements

---

SFELIST, *ELEM*, *Lab*

Lists the surface loads for elements.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Element at which surface load is to be listed. If ALL (or blank), list loads for all selected elements (**ESEL (p. 661)**). If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *ELEM*.

Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. See the **SFE (p. 1761)** command for labels.

Notes

The surface loads listed correspond to the current database values. The database is not updated for surface loads in POST1. Surface loads specified in tabular form, however, do list their values corresponding to the current results set in POST1.

For SURF151 or SURF152 elements with an extra node for radiation and/or convection calculations (KEYOPT(5) = 1), the bulk temperature listed is the temperature of the extra node. If the thermal solution does not converge, the extra node temperature is not available for listing.

Film effectiveness and free stream temperatures specified by the **SFE (p. 1761)** command (*Lab* = CONV) can only be listed by this command. The command lists film coefficients and bulk temperatures first and then film effectiveness and free stream temperatures below those values.

Distributed-Memory Parallel (DMP) Restriction

In a DMP analysis within the SOLUTION processor, **SFELIST (p. 1770)** support is not available for elements SURF151 and SURF152 when surface loading is applied via extra nodes (KEYOPT(5 > 0).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the command is issued under these circumstances, the resulting surface loads shown are not reliable.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Elements

Utility Menu>List>Loads>Surface Loads>On Picked Elems

SFFUN, *Lab*, *Par*, *Par2*

Specifies a varying surface load.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*. Issue **SFFUN (p. 1771)**,STATUS to list current command settings.

Discipline	Surface Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
Electric	CHRGS	surface charge density
Diffusion	DFLUX	Diffusion flux

1. Thermal labels CONV and HFLUX are mutually exclusive.

Par

Parameter containing list of surface load values. If *Lab* = CONV, values are typically the film coefficients and *Par2* values (below) are typically the bulk temperatures.

Par2

Parameter containing list of second surface load values (if any). If *Lab* = CONV, the *Par2* values are typically the bulk temperatures. *Par2* is not used for other surface load labels.

Notes

Specifies a surface load "function" to be used when the **SF (p. 1733)** or **SFE (p. 1761)** command is issued. The function is supplied through an array parameter vector which contains nodal surface load values. Node numbers are implied from the sequential location in the array parameter. For example, a value in location 11 applies to node 11. The element faces are determined from the implied list of nodes when the **SF (p. 1733)** or **SFE (p. 1761)** command is issued. Zero values should be supplied for nodes that

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

have no load. A tapered load value may be applied over the element face. These loads are in addition to any loads that are also specified with the **SF (p. 1733)** or **SFE (p. 1761)** commands. Issue **SFFUN (p. 1771)** (with blank remaining fields) to remove this specification. Issue **SFFUN (p. 1771)**,STATUS to list current settings.

Starting array element numbers must be defined for each array parameter vector. For example, **SFFUN (p. 1771)**,CONV,A(1,1),A(1,2) reads the first and second columns of array A (starting with the first array element of each column) and associates the values with the nodes. Operations continue on successive column array elements until the end of the column. Another example to show the order of the commands:

```
*dim,nodepres,array,2
nodepres(1)=11,12
/prep7
et,1,42
n,1
n,2,1
n,3,1,1
n,4,,1
e,1,2,3,4
sfe,1,1,pres,1,3
sfelist ! expected answer: 3 at both nodes 1 and 2
sfedel,all,pres,all
sffun,pres, nodepres(1)
sfe,1,1,pres,1,5
sfelist ! expected answer: 5+11=16 at node 1, 5+12=17 at node 2
fini
```

SFFUN (p. 1771) does not work for tabular boundary conditions.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>For Surface Ld>Node Function

Main Menu>Solution>Define Loads>Settings>For Surface Ld>Node Function

---

SFGRAD, *Lab*, *SLKCN*, *Sldir*, *SLZER*, *SLOPE*

Specifies a gradient (slope) for surface loads.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Surface Load Label	Label Description
Structural	PRES	pressure (real pressure only)
Thermal	CONV[1]	convection (bulk temperatures only)
	HFLUX[1]	heat flux
Electric	CHRGS	surface charge density
Diffusion	DFLUX	Diffusion flux

1. Thermal labels CONV and HFLUX are mutually exclusive.

SLKCN

Reference number of slope coordinate system (used with *Sldir* and *SLZER* to determine COORD). Defaults to 0 (the global Cartesian coordinate system).

Sldir

Slope direction in coordinate system *SLKCN*:

X

Slope is along X direction (default). Interpreted as R direction for non-Cartesian coordinate systems.

Y

Slope is along Y direction. Interpreted as θ direction for non-Cartesian coordinate systems.

Z

Slope is along Z direction. Interpreted as Φ direction for spherical or toroidal coordinate systems.

SLZER

Coordinate location (degrees for angular input) where slope contribution is zero (CVALUE = VALUE). Allows the slope contribution to be shifted along the slope direction. For angular input, *SLZER* should be between ±180° if the singularity (**CSCIR (p. 436)**) is at 180° and should be between 0° and 360° if the singularity is at 0°.

SLOPE

Slope value (load per unit length or per degree).

Notes

Specifies a gradient (slope) for surface loads. All surface loads issued with the **SF (p. 1733)**, **SFE (p. 1761)**, **SFL (p. 1774)**, or **SFA (p. 1739)** commands while this specification is active will have this gradient applied

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(for complex pressures, only the real component will be affected; for convections, only the bulk temperature will be affected). The load value, CVALUE, calculated at each node is:

CVALUE = VALUE + (*SLOPE* X (COORD-*SLZER*))

where VALUE is the load value specified on the subsequent **SF (p. 1733)**, **SFE (p. 1761)**, **SFL (p. 1774)**, or **SFA (p. 1739)** commands and COORD is the coordinate value (in the *Sldir* direction of coordinate system *SLKCN*) of the node. Only one **SFGRAD (p. 1772)** specification may be active at a time (repeated use of this command replaces the previous specification with the new specification). Issue **SFGRAD (p. 1772)** (with blank fields) to remove the specification. Issue **SFGRAD (p. 1772)**,STAT to show the current command status. The **SFGRAD (p. 1772)** specification (if active) is removed when the **LSREAD (p. 1083)** (if any) command is issued.

SFGRAD (p. 1772) does not work for tabular boundary conditions.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>For Surface Ld>Gradient

Main Menu>Solution>Define Loads>Settings>For Surface Ld>Gradient

---

SFL, *Line*, *Lab*, *VALI*, *VALJ*, *VAL2I*, *VAL2J*

Specifies surface loads on lines of an area.

SOLUTION (p. 38): Solid Surface Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Line

Line to which surface load applies. If ALL, apply load to all selected lines (**LSEL (p. 1074)**). If *Line* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *Line*.

Lab

Valid surface load label. Load labels are listed under "Surface Loads" in the input table for each element type in the *Element Reference*.

Discipline	Body Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Body Load Label	Label Description
	RAD	radiation
	RDSF	surface-to-surface radiation
Fluid	FSI[2]	fluid-structure interaction flag
	IMPD	impedance boundary flag
Magnetic	MXWF	Maxwell force flag
Infinite element	INF	Exterior surface flag for INFIN110 and INFIN111
Electric	CHRGS	surface charge density
	MXWF	Maxwell electrostatic force flag
Field-surface interface	FSIN	field-surface interface number

1. Thermal labels CONV and HFLUX are mutually exclusive.
2. For an acoustic analysis, apply the fluid-structure interaction flag (Label = FSI) to only the FLUID129 or FLUID130 elements.

VALI*, *VALJ

Surface load values at the first keypoint (*VALI*) and at the second keypoint (*VALJ*) of the line, or table name for specifying tabular boundary conditions. If *VALJ* is blank, it defaults to *VALI*. If *VALJ* is zero, a zero is used.

If *Lab* = CONV:

VALI and *VALJ* are the film coefficients and *VAL2I* and *VAL2J* are the bulk temperatures.

To specify a table, enclose the table name in percent signs (%), e.g., %*tabname*%. (Issue ***DIM (p. 530)** to define a table.)

If *Lab* = CONV and *VALI* = -*N*, the film coefficient may be a function of temperature and is determined from the HF property table for material *N* (**MP (p. 1160)**). (See **SCOPT (p. 1649)** for a way to override this option and use -*N* as the film coefficient.)

If **KBC (p. 970)**,0 has been issued for ramped loads, it affects only the bulk temperatures, *VAL2I* and *VAL2J*, and the film coefficient specification, *VALI* and *VALJ*, are unaffected.

If *Lab* = RAD, *VALI* and *VALJ* values are surface emissivities and *VAL2I* and *VAL2J* are ambient temperatures. The temperature used to evaluate the film coefficient is usually the average between the bulk and wall temperatures, but may be user-defined for some elements.

If *Lab* = RDSF, *VALI* is the emissivity value. If *VALI* = -*N*, the emissivity may be a function of the temperature and is determined from the EMISS property table for material *N* (**MP (p. 1160)**).

If *Lab* = FSIN in a unidirectional Mechanical APDL-to-CFX analysis, *VALJ* is the surface interface number (not available from within the GUI) and *VALI* is not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VAL2I, VAL2J

Second surface load values (if any). If *VAL2J* is blank, it defaults to *VAL2I*. If *VAL2J* is zero, a zero is used. To specify a table (*Lab* = CONV), enclose the table name in percent signs (%), for example, %*tabname*%. Use the ***DIM (p. 530)** command to define a table.

If *Lab* = CONV:

VAL2I and *VAL2J* are the bulk temperatures.

If **KBC (p. 970)**,0 has been issued for ramped loads, the bulk temperature is ramped from the value defined by **TUNIF (p. 2027)** to the value specified by *VALI* and *VALJ* for the first loadstep. If tabular boundary conditions are defined, the **KBC (p. 970)** command is ignored and tabular values are used for the bulk temperatures.

If *Lab* = RAD, *VAL2I* and *VAL2J* are the ambient temperatures.

If *Lab* = RDSF, *VAL2I* is the enclosure number. Radiation will occur between surfaces flagged with the same enclosure numbers. If the enclosure is open, radiation will occur to the ambient.

VAL2I and *VAL2J* are not used for other surface load labels.

Notes

Specifies surface loads on the selected lines of area regions. The lines represent either the edges of area elements or axisymmetric shell elements themselves. Surface loads may be transferred from lines to elements via **SFTRAN (p. 1782)** or **SBCTRAN (p. 1647)**. See **SFE (p. 1761)** for a description of surface loads. Loads input on this command may be tapered. See **SFGRAD (p. 1772)** for an alternate tapered load capability.

You can specify a table name only when using structural (PRES) and thermal (CONV [film coefficient and/or bulk temperature], HFLUX), and surface emissivity and ambient temperature (RAD) surface load labels. *VALJ* and *VAL2J* are ignored for tabular boundary conditions.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via **LVSCALE (p. 1094)**.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppInfinite>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Flag>AppMaxwell>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Field Surface Intr>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid/ANSYS>Impedance>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pressure>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Convection>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Heat Flux>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Radiation>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Excitation>AppSurfChar>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppInfinite>On Lines

Main Menu>Solution>Define Loads>Apply>Electric>Flag>AppMaxwell>On Lines

Main Menu>Solution>Define Loads>Apply>Field Surface Intr>On Lines

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Field Surface>On Lines

Main Menu>Solution>Define Loads>Apply>Fluid/ANSYS>Impedance>On Lines

Main Menu>Solution>Define Loads>Apply>Magnetic>Flag>AppInfinite>On Lines

Main Menu>Solution>Define Loads>Apply>Magnetic>Other>AppMaxwell>On Lines

Main Menu>Solution>Define Loads>Apply>Structural>Pressure>On Lines

Main Menu>Solution>Define Loads>Apply>Thermal>Convection>On Lines

Main Menu>Solution>Define Loads>Apply>Thermal>Heat Flux>On Lines

Main Menu>Solution>Define Loads>Apply>Thermal>Radiation>On Lines

SFLDELE, *LINE*, *Lab*

Deletes surface loads from lines.

SOLUTION (p. 38): Solid Surface Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

Line to which surface load deletion applies. If ALL, delete load from all selected lines (**LSEL (p. 1074)**). If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

Lab

Valid surface load label. If ALL, use all appropriate labels. See the **SFL (p. 1774)** command for labels.

Notes

Deletes surface loads (and all corresponding finite element loads) from selected lines.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Surface Ld>On All Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppInfinite>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Electric>Flag>AppMaxwell>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Field Surface Intr>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Fluid/ANSYS>Impedance>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Structural>Pressure>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Ambient Rad>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Convection>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Heat Flux>On Lines

Main Menu>Preprocessor>Loads>Define Loads>Delete>Thermal>Radiation>On Lines

Main Menu>Solution>Define Loads>Delete>All Load Data>All Surface Ld>On All Lines

Main Menu>Solution>Define Loads>Delete>Electric>Excitation>AppSurfChar>On Lines

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppInfinite>On Lines

Main Menu>Solution>Define Loads>Delete>Electric>Flag>AppMaxwell>On Lines

Main Menu>Solution>Define Loads>Delete>Field Surface Intr>On Lines

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Field Surface>On Lines

Main Menu>Solution>Define Loads>Delete>Fluid/ANSYS>Impedance>On Lines

Main Menu>Solution>Define Loads>Delete>Magnetic>Flag>AppInfinite>On Lines

Main Menu>Solution>Define Loads>Delete>Magnetic>Other>AppMaxwell>On Lines

Main Menu>Solution>Define Loads>Delete>Structural>Pressure>On Lines

Main Menu>Solution>Define Loads>Delete>Thermal>Ambient Rad>On Lines

Main Menu>Solution>Define Loads>Delete>Thermal>Convection>On Lines

Main Menu>Solution>Define Loads>Delete>Thermal>Heat Flux>On Lines

Main Menu>Solution>Define Loads>Delete>Thermal>Radiation>On Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SFLEX, *FFAX*, *FFBY*, *FFBZ*, *FFT0*, *FFTSY*, *FFTSZ*

Sets flexibility factors for the currently defined pipe element section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FFAX

Factor to increase axial flexibility. The default value is 1.0.

FFBY

Factor to increase bending flexibility about element y axis (bending in the element x-z plane). The default value is 1.0.

FFBZ

Factor to increase bending flexibility about element z axis (bending in the element x-y plane). The default value is *FFBY*.

FFT0

Factor to increase torsional flexibility. The default value is 1.0.

FFTSY

Factor to increase transverse shear flexibility in the element x-z plane. The default value is 1.0.

FFTSZ

Factor to increase transverse shear flexibility in the element x-y plane. The default value is *FFTSY*.

Notes

The **SFLEX (p. 1779)** command sets section-flexibility factors for sections used by pipe elements.

To increase *stiffness*, use a flexibility factor of less than 1.0.

The *FFBY* and *FFTSY* arguments affect motion in the element x-z plane, and the *FFBZ* and *FFTSZ* arguments affect motion in the element x-y plane. For stout pipe structures with low slenderness ratios, set *both* FFBY and FFTSY--and/or *both* FFBZ and FFTSZ (the related bending and transverse shear factors)--to the same value to obtain the expected flexibility effect.

When issued, the **SFLEX (p. 1779)** command applies to the pipe section most recently defined via the **SECTYPE (p. 1697)** command.

SFLEX (p. 1779) is valid only for linear material properties and small strain analyses. The command does not support offsets, temperature loading, or initial state loading. While the resulting displacements and reactions are valid, the stresses may not be valid.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Sections>Pipe>Flexibility Factors

SFLIST, *NODE*, *Lab*

Lists surface loads.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node at which surface load is to be listed. If ALL (or blank), list for all selected nodes (**NSEL (p. 1266)**). If *NODE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *NODE*.

Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. See the **SF (p. 1733)** command for labels

Notes

Lists the surface loads as applied with the **SF (p. 1733)** command. Loads are listed only for the specified nodes on external faces of selected area and volume elements. Use **SFELIST (p. 1770)** for line elements. The surface loads listed correspond to the current database values. The database is not updated for surface loads in POST1. Surface loads specified in tabular form, however, do list their values corresponding to the current results set in POST1.

For SURF151 or SURF152 elements with an extra node for radiation and/or convection calculations (KEYOPT(5) = 1), the bulk temperature listed is the temperature of the extra node. If the thermal solution does not converge, the extra node temperature is not available for listing.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Nodes

Utility Menu>List>Loads>Surface Loads>On Picked Nodes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SFLLIST, *LINE*, *Lab*

Lists the surface loads for lines.

SOLUTION (p. 38): Solid Surface Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LINE

Line at which surface load is to be listed. If ALL (or blank), list for all selected lines (**LSEL (p. 1074)**). If *LINE* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may be substituted for *LINE*.

Lab

Valid surface load label. If ALL (or blank), use all appropriate labels. See the **SFL (p. 1774)** command for labels.

Notes

Lists the surface loads for the specified line.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Loads>Surface Loads>On All Lines

Utility Menu>List>Loads>Surface Loads>On Picked Lines

SFSCALE, *Lab*, *FACT*, *FACT2*

Scales surface loads on elements.

SOLUTION (p. 38): FE Surface Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid surface load label. If ALL, use all appropriate labels.

Discipline	Surface Load Label	Label Description
Structural	PRES	pressure
Thermal	CONV[1]	convection
	HFLUX[1]	heat flux
Substructure	SELV	load vector number
Electric	CHRGS	surface charge density

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discipline	Surface Load Label	Label Description
Diffusion	DFLUX	Diffusion flux

1. Thermal labels CONV and HFLUX are mutually exclusive.

FACT

Scale factor for the first surface load value. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

FACT2

Scale factor for the second surface load value. Zero (or blank) defaults to 1.0. Use a small number for a zero scale factor.

Notes

Scales surface loads (pressure, convection, etc.) in the database on the selected elements. Surface loads are applied with the **SF (p. 1733)**, **SFE (p. 1761)**, or **SFBEAM (p. 1746)** commands. Issue the **SFELIST (p. 1770)** command to list the surface loads. Solid model boundary conditions are not scaled by this command, but boundary conditions on the FE model are scaled.

Note:

Such scaled FE boundary conditions may still be overwritten by unscaled solid model boundary conditions if a subsequent boundary condition transfer occurs.

SFSCALE (p. 1781) does not work for tabular boundary conditions.

This command is also valid in PREP7 and in the **/MAP (p. 1107)** processor.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Scale FE Loads>Surface Loads

Main Menu>Solution>Define Loads>Operate>Scale FE Loads>Surface Loads

SFTRAN

Transfer the solid model surface loads to the finite element model.

SOLUTION (p. 38): Solid Surface Loads (p. 44)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Surface loads are transferred only from selected lines and areas to all selected elements. The **SFTRAN (p. 1782)** operation is also done if the **SBCTRAN (p. 1647)** command is issued or automatically done upon initiation of the solution calculations (**SOLVE (p. 1822)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Operate>Transfer to FE>Surface Loads

Main Menu>Solution>Define Loads>Operate>Transfer to FE>Surface Loads

---

/SHADE, *WN*, *Type*

Defines the type of surface shading used with Z-buffering.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

Type

Shading type:

FACET or 0

Facet shading (one color per area face).

GOURAUD or 1

Gouraud smooth shading (smooth variation of color based on interpolated vertex colors) (default).

PHONG or 2

Phong smooth shading (smooth variation of color based on interpolated vertex normals).

Command Default

Facet shading.

Notes

Defines the type of surface shading used on area, volume, and PowerGraphics (**/GRAPHICS (p. 866)**,POWER) displays when software Z-buffering is enabled (**/TYPE (p. 2034)**). This command is only functional for 2D display devices.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHELL, *Loc*

Selects a shell element or shell layer location for results output.

POST1 (p. 51): Controls (p. 51)
POST26 (p. 58): Controls (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Loc

Location within shell element (or layer) to obtain stress results:

TOP

Top of shell element (or layer) (default).

MID

Middle of shell element (or layer). The default method averages the TOP and BOT values to obtain a mid value. Setting KEYOPT(8) = 2 for SHELL181, SHELL208, SHELL209, SHELL281, and ELBOW290 uses MID results obtained directly from the results file.

BOT

Bottom of shell element (or layer).

Command Default

Shell element (or layer) top location.

Notes

Selects the location within a shell element (or a shell layer) for results output (nodal stresses, strains, etc.). Applies to POST1 selects, sorts, and output (**NSEL (p. 1266)**, **NSORT (p. 1283)**, **PRNSOL (p. 1505)**, **PLNSOL (p. 1425)**, **PRPATH (p. 1517)**, **PLPATH (p. 1439)**, etc.), and is used for storage with the POST26 **ESOL (p. 672)** command. For example, **SHELL (p. 1784)**,TOP causes item S of the POST1 **PRNSOL (p. 1505)** command or the POST26 **ESOL (p. 672)** command to be the stresses at the top of the shell elements. For layered shell elements, use the **LAYER (p. 1015)** (POST1) or **LAYERP26 (p. 1016)** (POST26) command to select the layer. The **SHELL (p. 1784)** command does not apply to the layered thermal shell elements, SHELL131 and SHELL132.

For PowerGraphics (**/GRAPHICS (p. 866)**,POWER), the **SHELL (p. 1784)**,MID command affects both the printed output and the displayed results, while the **SHELL (p. 1784)** (TOP or BOT) command prints and displays both the top and bottom layers simultaneously. Note that **/CYCEXPAND (p. 454)**,ON automatically turns on PowerGraphics; however, for cyclic mode-superposition harmonic postprocessing (**CYCFILES (p. 457)**), the **SHELL (p. 1784)** command prints and displays only the requested layer.

In POST26, the **ESOL (p. 672)** data stored is based on the active **SHELL (p. 1784)** specification at the time the data is stored. To store data at various specifications (for example, stresses at the top and bottom locations), issue a **STORE (p. 1867)** command before each new specification.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>Options for Outp

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

Utility Menu>List>Results>Options

/**SHOW**,*Fname*,*Option*,*VECT*,*NCPL*

Specifies the device and other parameters for graphics displays.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

Device name, file name, or keyword, as listed below:

<device name>

Any valid graphics display device name (for example, X11, 3D etc.). Defaults to X11 for most systems. See Getting Started with Graphics in the *Basic Analysis Guide* for details. A device name must be defined before activating the Graphical User Interface (GUI). Once the GUI is activated, the device name cannot be changed for that session, except for switching between X11 and X11C.

TERM

Graphics displays are switched back to the last-specified device name.

CLOSE

This option purges the graphics file buffer. The CLOSE option should be issued any time you are changing graphics devices or file output types during a session. Graphics displays are switched back to the last-specified device name, and any open graphics files are closed. The CLOSE option is similar to the TERM option, however, with the CLOSE option, another process can access the data in the graphics file. The CLOSE option causes graphics file buffers to be flushed to the graphics file.

OFF

Graphics display requests are ignored.

(blank)

If blank in interactive mode, graphics will be displayed on screen as requested by display commands (no file written).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

JPEG

Creates JPEG files that are named Jobname*nnn*.jpg, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, Jobname000.jpg, Jobname001.jpg, Jobname002.jpg, and so on. Ignores the *Ext* field.

TIFF

Creates tagged image format files that are named Jobname*nnn*.tif, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, Jobname000.tif, Jobname001.tif, Jobname002.tif, and so on. This value for the *Fname* argument ignores the *Ext* field. (See the **TIFF (p. 1999)** command for options.)

PNG

Creates PNG (Portable Network Graphics) files that are named Jobname*nnn*.png, where *nnn* is a numeric value that is incremented by one as each additional file is created; that is, Jobname000.png, Jobname001.png, Jobname002.png, and so on. This value for the *Fname* argument ignores the *Ext* field. (See the **PNGR (p. 1455)** command for options.)

VRML

Creates Virtual Reality Meta Language files named Jobname000.wrl that can be displayed on 3D Internet web browsers. Ignores the *Ext* and *NCPL* fields.

Option

Assign a file name extension or specify reverse video output:

Ext

File name extension (eight-character maximum).

REV

Reverse background/image (black/white) colors. Valid with *Fname* = PNG (recommended), JPEG, and TIFF. This option is ignored if a previously specified color map table (**/CMAP (p. 359)** or **/RGB (p. 1608)**) is in effect.

VECT

Specifies raster or vector display mode. This affects area, volume, and element displays, as well as geometric results displays such as contour plots. See the **/DEVICE (p. 523)** command for an alternate way to toggle between raster and vector mode. Changing *VECT* also resets the **/TYPE (p. 2034)** command to its default.

0

Raster display (color filled entities; default)

1

Vector display (outlined entities; that is, "wireframe")

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NCPL

Sets the number of color planes (4 to 8). Default is device-dependent. *NCPL* is not supported by all graphics devices.

Command Default

For interactive runs, display is shown on the screen; for batch runs, display commands are ignored (graphics file not written).

Notes

Specifies the device to be used for graphics displays, and specifies other graphics display parameters. Display may be shown at the time of generation (for interactive runs at a graphics display terminal) or diverted to a file. Issue **/PSTATUS (p. 1550)** for display status.

Batch runs do not have access to the fonts available on your system. The Courier and Helvetica font files used for JPEG, PNG and TIFF batch output are copyrighted by Adobe Systems Inc. and Digital Equipment Corp. Permission to use these trademarks is hereby granted only in association with the images described above. Batch run JPEG output is produced at the default quality index value of 75, unless specified otherwise.

Interactive displays default to eight color planes (*NCPL* = 8) for most monitors, while graph file output defaults to eight color planes for VRML output, and four color planes for JPEG, PNG, and TIFF.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Device Options

Utility Menu>PlotCtrls>Redirect Plots>To Screen

SHPP, *Lab*, *VALUE1*, *VALUE2*

Controls element shape checking.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Shape-checking option. (When *Lab* = WARN, STATUS, SUMMARY, or DEFAULT, the remaining arguments are ignored.)

ON

Activates element shape-checking. New elements, regardless of how they are created, are tested against existing warning and error limits. (The existing limits may be the default limits, or previously modified limits.) Elements that violate error limits produce *error* messages and either (a) cause a meshing failure, or (b) for element creation or storage other than

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AMESH (p. 121) or **VMESH (p. 2140)**, *are not stored*. Elements that violate warning limits produce *warning* messages. If shape-checking was previously turned off (**SHPP (p. 1787)**,OFF) and you turn it on, existing elements are marked as untested; use the **CHECK (p. 333)** command to retest them. With this option, you may also specify a value for *VALUE1* to turn individual shape tests on. If you do not specify a value for *VALUE1*, all shape tests are turned on.

WARN

Activates element shape-checking; however, in contrast to **SHPP (p. 1787)**,ON, elements that violate error limits do not cause either a meshing or element storage failure. Instead, they produce *warning* messages to notify you that error limits have been violated. This option does not alter current shape parameter limits. Since the default shape parameter error limits are set to allow almost any usable element, the elements this option allows, which would otherwise be forbidden, are likely to be *very poorly shaped*.

OFF

Deactivates element shape-checking. This setting does not alter current shape parameter limits. Use of this option is risky, since poorly shaped elements can lead to analysis results that are less accurate than would otherwise be expected for a given mesh density. With this option, you may also specify a value for *VALUE1* to turn individual shape tests off. If you do not specify a value for *VALUE1*, all element shape tests are turned off.

SILENT

Determines whether element shape-checking runs in silent mode. In silent mode, Mechanical APDL checks elements without issuing warnings, *with the exception of* the generic warnings that it issues at solution. With this option, you must also specify a value for *VALUE1* (During the execution of certain commands, Mechanical APDL automatically runs element shape-checking in silent mode, then internally summarizes the shape test results for all of the new or modified elements. Mechanical APDL does this when it executes any of the following commands: **AGEN (p. 107)**, **AMESH (p. 121)**, **AREFINE (p. 178)**, **ARSYM (p. 184)**, **ATRAN (p. 205)**, **CDREAD (p. 291)**, **EGEN (p. 603)**, **ENGEN (p. 642)**, **ENSYM (p. 644)**, **EREAD (p. 653)**, **EREFINE (p. 654)**, **ESYM (p. 684)**, **ET (p. 686)**, **FVMESH (p. 784)**, **KREFINE (p. 990)**, **LREFINE (p. 1062)**, **NREFINE (p. 1257)**, **TIMP (p. 2003)**, **VEXT (p. 2091)**, **VGEN (p. 2116)**, **VIMP (p. 2132)**, **VMESH (p. 2140)**, **VOFFST (p. 2140)**, **VROTAT (p. 2155)**, **VSWEEP (p. 2168)**, **VSYMM (p. 2169)**, and **VTRAN (p. 2171)**.)

STATUS

Lists the shape parameter limits currently in effect, along with status information about element shape-checking (for example, whether any individual shape tests are off, whether any of the shape parameter limits have been modified, and so on).

SUMMARY

Lists a summary of element shape test results for all selected elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEFAULT

Resets element shape parameter limits to their default values. Also, if any individual tests were turned off, turns them back on. (The **SHPP (p. 1787)**,DEFAULT command may be useful if any parameter limits were previously altered by using the MODIFY option.)

OBJECT

Determines whether element shape test results data is *stored in memory*. When this option is turned on, an "object" is created for storing test results in memory. When this option is turned off, no object is created and no data is stored; thus, any operation that requires shape parameters for an existing element (such as use of the **CHECK (p. 333)** command) causes the shape parameters to be recomputed. (Note the distinction between storing the data in memory and storing it in the database; regardless of whether this option is turned on or off, no element shape test results data will be stored in the database. The element shape parameter object is deleted automatically before any solution.) This setting is independent of shape-checking status, with one exception--if shape-checking is turned off (**SHPP (p. 1787)**,OFF), the object is not created. Keep in mind that recomputing shape parameters is more computationally expensive than retrieving them from the object. With this option, you must also specify a value for the *VALUE1* argument; the *VALUE2* argument is ignored.

LSTET

Determines, for Jacobian ratio tests, whether sampling is done at integration points or at corner nodes. When this option is turned on, sampling is done at integration points, and the default limits for h-element Jacobian ratios are a warning tolerance of 10 and an error tolerance of 40. When this option is turned off, sampling is done at corner nodes, and the corresponding default limits are a warning tolerance of 30 and an error tolerance of 1000. Sampling at the integration points (option on) results in a lower Jacobian ratio, but that ratio is also subjected to a more restrictive error limit. Some elements that have passed the integration point sampling criterion, have failed the corner mode sampling criterion. Because of this, use integration point sampling only for simple linear analyses. For other types of analyses (e.g., nonlinear, electromagnetic), use sampling at corner nodes, which is the more conservative approach. With this option, you must also specify a value for the *VALUE1* argument; the *VALUE2* argument is ignored.

MODIFY

Indicates that you want to respecify a shape parameter limit. With this option, you must also specify values for the *VALUE1* and *VALUE2* arguments.

FLAT

Determines the warning and error limits used to test elements that may exhibit nonzero/nonconstant Z coordinates. With this option, you must also specify values for the *VALUE1* and/or *VALUE2* arguments.

VALUE1

Valid for the ON, OFF, SILENT, OBJECT, LSTET, MODIFY, and FLAT options only. When *Lab* = ON or OFF, use *VALUE1* to individually control (that is, turn off or turn on) specific element shape tests. Thus, *VALUE1* can be ASPECT (aspect ratio tests), PARAL (deviation from parallelism of opposite

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

edges tests), MAXANG (maximum corner angle tests), JACRAT (Jacobian ratio tests), WARP (warping factor tests), or ALL (all tests). When *Lab* = SILENT, *VALUE1* can be ON (to turn silent mode on) or OFF (to turn silent mode off). When *Lab* = OBJECT, *VALUE1* can be either 1, YES, or ON to turn on storage of element shape test data (the default); or it can be 0, NO, or OFF to turn off storage of element shape test data (delete the data and recompute as necessary). When *Lab* = LSTET, *VALUE1* can be either 1, YES, or ON to choose Jacobian sampling at integration points; or it can be 0, NO, or OFF to choose Jacobian sampling at nodes (the default). When *Lab* = MODIFY, *VALUE1* is the numeric location (within the shape parameter limit array) of the shape parameter limit to be modified. Locations are identified in the element shape-checking status listing (**SHPP (p. 1787)**,STATUS). For more information, see the examples in the **Notes** section. When *Lab* = FLAT, *VALUE1* is the warning limit for XY element constant Z sets performed at **CHECK (p. 333)** or **SOLVE (p. 1822)**. The default is 1.0e-8.

VALUE2

Valid for the MODIFY and FLAT options only. When *Lab* = MODIFY, specifies the new limit for the shape parameter that is in the location indicated by the *VALUE1* argument. See the examples in the **Notes** section. When *Lab* = FLAT, *VALUE2* is the error limit. The default is 1.0e-2.

Command Default

All shape-checking tests are enabled (**SHPP (p. 1787)**,ON,ALL) with default shape parameter limits. Silent mode is off. Memory object storage of element shape parameters is on.

Notes

The following examples illustrate how to use the **SHPP (p. 1787)**,MODIFY,*VALUE1*,*VALUE2* command to respecify shape parameter limits. Assume that you issued the **SHPP (p. 1787)**,STATUS command, and you received the output below:

```
ASPECT RATIO (EXCEPT EMAG)

   QUAD OR TRIANGLE ELEMENT OR FACE
        WARNING TOLERANCE ( 1) =   20.00000
        ERROR TOLERANCE   ( 2) =  1000000.
                    .
                    .
                    .
MAXIMUM CORNER ANGLE IN DEGREES (EXCEPT OR EMAG)
   TRIANGLE ELEMENT OR FACE
        WARNING TOLERANCE (15) =   165.0000
        ERROR TOLERANCE   (16) =   179.9000
```

Notice that in the sample output, the *warning* tolerance for aspect ratios is set to 20. Now assume that you want to "loosen" this shape parameter limit so that it is less restrictive. To allow elements with aspect ratios of up to 500 without causing warning messages, you would issue this command:

```
SHPP (p. 1787),MODIFY,1,500
```

Also notice that each shape parameter's numeric location within the shape parameter limit array appears in the sample output within parentheses. For example, the numeric location of the aspect ratio shape parameter (for warning tolerance) is 1, which is why "1" is specified for the *VALUE1* argument in the example command above.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Now notice that the sample output indicates that any triangle element with an internal angle that is greater than 179.9 degrees will produce an *error* message. Suppose that you want to "tighten" this shape parameter limit, so that it is more restrictive. To cause any triangle or tetrahedron with an internal angle greater than 170 degrees to produce an error message, you would issue this command:

```
SHPP (p. 1787),MODIFY,16,170
```

The existence of badly shaped elements in a model may lead to certain computational errors that can cause your system to terminate during solution. Therefore, you run the risk of a system abort during solution any time that you disable element shape-checking entirely, run shape-checking in warning-only mode, disable individual shape-checks, or loosen shape-parameter limits.

Changing any shape parameter limit marks all existing elements as untested; use the **CHECK (p. 333)** command to retest them.

For more information about element shape-checking, see Meshing Your Solid Model.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Checking Ctrls>Shape Checking

Main Menu>Preprocessor>Checking Ctrls>Toggle Checks

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Element Shape Checking>Shape Checking

Main Menu>Solution>Manual Rezoning>Create Remesh Zone(s)>Mesh Controls>Element Shape Checking>Toggle Checks

---

SHSD,*RID*,*Action*,*CHCH_Opt*,*CGAP*,*CPEN*

Creates or deletes a shell-solid interface to be used in shell-to-solid assemblies.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RID

The real constant set ID that identifies the contact pair on which a shell-to-solid assembly is defined. If ALL, all selected contact pairs will be considered for assembly.

Action

Action to be performed:

EDGE

Create virtual shell elements based on the shell edge (default).

SURFACE

Create virtual shell elements based on the solid element surface.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELETE

Delete the nodes and elements created during a previous execution of **SHSD (p. 1791)** for the real constant set identified by *RID*.

CNCH_Opt

CNCHECK (p. 382) option to close initial penetration or gap:

NONE

No adjustment of initial penetration or gap (default).

ADJUST

Use **CNCHECK (p. 382)**,ADJUST logic internally to close the initial gap or penetration (see the **CNCHECK (p. 382)** command for detail). *CGAP* or *CPEN* must be specified.

CGAP

Control parameter for opening gap; must be greater than or equal to zero. Close the gap if the gap distance is smaller than the *CGAP* value. *CGAP* defaults to 0.25*PINB (where PINB is the pinball radius) for bonded and no-separation contact. Otherwise, it defaults to the value of real constant ICONT.

CPEN

Control parameter for initial penetration; must be greater than or equal to zero. Close the penetration if the penetration distance is smaller than the *CPEN* value. *CPEN* defaults to 0.25*PINB (where PINB is the pinball radius) for any type of interface behavior (either bonded or standard contact).

Notes

The **SHSD (p. 1791)** command creates a shell-solid interface to be used in shell-to-solid assemblies, or deletes a previously-created shell-solid interface. *Virtual* shell elements and additional CONTA175 or CONTA177 elements are created at the contact pair identified by *RID* when *Action* = EDGE or SURFACE. Set *Action* = DELETE to remove the generated nodes and elements at the contact pair identified by *RID*.

The **SHSD (p. 1791)** command is active only when the following element KEYOPTs of associated CONTA175 or CONTA177 element types are predefined:

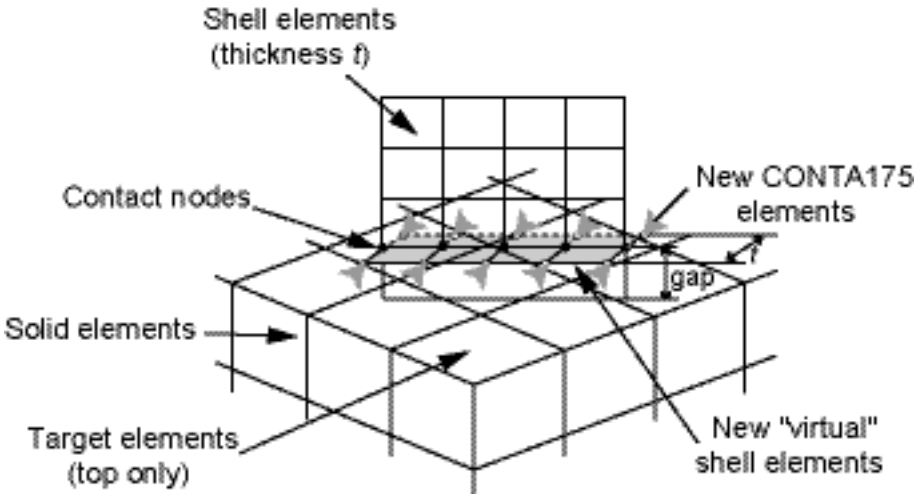
Element	KEYOPT	Detail
CONTA175 or CONTA177	KEYOPT(2) = 2	MPC algorithm
	KEYOPT(12) = 5, 6	Bonded contact
	KEYOPT(4) = 0	Contact normal perpendicular to target surface

When *ACTION* = EDGE, the virtual shell elements are built perpendicular to the pre-existing shell elements attached to the contact elements. They geometrically follow the contact interface edge and are built on both sides of this interface in such a way that each new shell element (SHELL181) has two nodes that belong to the associated pre-existing shell element in the shell edge. (See Figure 26: Virtual Shell Elements Following the Contact Interface Edge (p. 1793).) The width of the new shell elements is half the

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

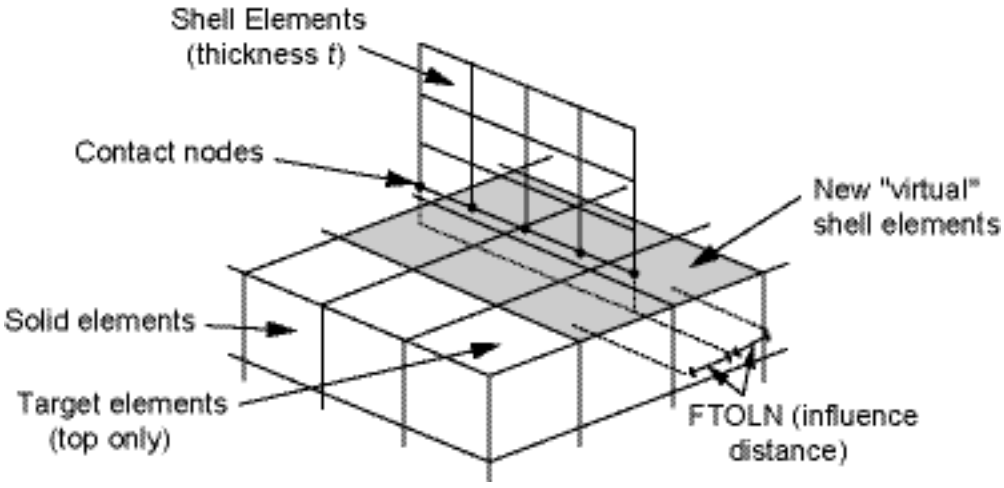
thickness of the pre-existing shell element. If CONTA175 is used at the shell nodes, the CONTA175 elements are then created at each node of the virtual shell elements where no CONTA175 element exists. If CONTA177 is used at the shell edge, new CONTA174 elements are created and overlayed on the virtual shell. The new contact elements are identified by the same contact pair ID as the pre-existing contact elements. The virtual shell elements are assigned the next available element type number and material number.

Figure 26: Virtual Shell Elements Following the Contact Interface Edge



When *ACTION* = SURFACE, the virtual shell elements (SHELL181 - low order; SHELL281 - high order) overlap the existing low or high order target elements identified with the *RID* argument, and share their nodes. Only those target elements close enough to the contact interface (identified using the PINB real constant) are overlapped. The program uses the FTOLN real constant (defaults to half the shell element thickness) to define an influence distance. The associated virtual shell elements are created only for target elements that lie partially inside the influence distance region (see Figure 27: Virtual Shell Elements Overlapping Target Elements (p. 1793)).

Figure 27: Virtual Shell Elements Overlapping Target Elements



For the bonded always option (KEYOPT(12) = 5), any contact node inside the pinball region (gap < PINB) is included in the KEYOPT(5) = 2 process. A relatively small PINB value may be used to prevent false contact. PINB defaults to 25% of the contact depth for small deformation analyses.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

For the bonded initial option (KEYOPT(12) = 6), only those contact nodes which initially lie inside the adjustment zone (gap < ICONT) are always included in the KEYOPT(5) = 2 process. ICONT defaults to 5% of the contact depth.

For both processes, the new nodes and elements are stored in internally-named components. The internal naming convention is based on the real constant set ID specified by *RID*, as illustrated in the following table.

Nodes	SHSD_ND_*RID*
Contact Elements	SHSD_CN_*RID*
Shell Elements	SHSD_SH_*RID*

Issuing **SHSD (p. 1791)**,*RID*,DELETE deletes components based on their generated names. *Only components whose names match the internal naming convention will be deleted.*

Caution:

Do not rename or manually delete generated components. Use the **SHSD (p. 1791)** command to delete generated components.

Renaming or manually deleting generated components will cause these components to be ignored when **SHSD (p. 1791)**,*RID*,DELETE is executed and when the program searches for these components to verify if **SHSD (p. 1791)**,*RID*,EDGE or SURFACE can be safely executed. Manually renaming or deleting generated components and reissuing **SHSD (p. 1791)**,*RID*,EDGE or SURFACE may result in erroneous generation of virtual shell or contact elements.

SHSD (p. 1791) does not support assemblies that contain a preintegrated shell section (**SECTYPE (p. 1697)**,,GENS).

See Modeling a Shell-Solid Assembly in the *Contact Technology Guide* for more information.

Menu Paths

This command cannot be accessed from a menu.

/SHRINK, *RATIO*

Shrinks elements, lines, areas, and volumes for display clarity.

GRAPHICS (p. 16): Scaling (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RATIO

Shrinkage ratio (input as a decimal (0.0 to 0.5)). Defaults to 0.0 (no shrinkage). Values greater than 0.5 default to 0.1 (10% shrinkage).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Full size entities.

Notes

Shrinks the elements, lines, areas, and volumes so that adjacent entities are separated for clarity. Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

If only the common lines of non-coplanar faces are drawn (as per the **/EDGE (p. 594)** command), then this command is ignored.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Size and Shape

SLIST,*SFIRST*,*SLAST*,*SINC*,*Details*,*Type*

Summarizes the section properties for all defined sections in the current session.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SFIRST

First section ID to be summarized. Default = First available section in the database.

SLAST

Last section ID to be summarized. Default = Last available section in the database.

SINC

Increment of the section ID. Default = 1.

Details

Determines the content of the summarized information for beam, pipe, shell, and reinforcing sections.

BRIEF

For beams, lists only the section integrated properties (such as *Area*, *Iyy*, and *Iyz*). This option is the default.

For reinforcing, lists only the input reinforcing properties (such as material, cross-section area, fiber spacing, and input fiber location parameters).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FULL

For beams, lists the section integrated properties, as well as the section nodal coordinates, section cell connectivity information, and section cell integration point coordinates. For shells, the section stiffness (membrane, bending, membrane-bending coupling and transverse shear) are printed.

The shell section stiffness listed considers elastic behavior of materials at reference temperature only. The elements that use the section data may alter the transverse shear stiffness based on slenderness considerations (in addition to the shear correction factors shown).

Section stiffness terms listed via the FULL option do not include section offsets. The program accounts for section offsets during the solution phase of the analysis.

For predefined reinforcing sections used by the standard method, lists the complete information of reinforcing fibers or surfaces (including material ID, cross-section area, fiber spacing, orientation, and locations in natural coordinates). For predefined section reinforcing used by the mesh-independent method, lists section properties to generate the fibers or surfaces (including material ID, cross-section area, fiber spacing, and orientation) For reinforcing sections generated (**ERE-INF (p. 656)**) via the mesh-independent method, lists element ID, material ID, and locations in natural coordinates.

GROUP

If a section calls other sections, this option lists those sections too.

Type

The section type. Valid arguments are ALL (default) or any valid section type (**SECTYPE (p. 1697)**).

Notes

By default, the command lists information about all sections. You can limit the output to specific section types via the *Type* key.

When ocean loading is present, the command lists beam section properties used by ocean loading.

Example Output

Following is example output from the **SLIST (p. 1795)**,,,,BRIEF command for a rectangular beam section subtype (**SECTYPE (p. 1697)**,,BEAM,RECT):

```
LIST SECTION ID SETS     1 TO     1 BY     1

  SECTION ID NUMBER:         1
  BEAM SECTION TYPE:     Rectangle
  BEAM SECTION NAME IS:
  BEAM SECTION DATA SUMMARY:
   Area                =  6.0000
   Iyy                 =  4.5000
   Iyz                 = 0.11281E-15
   Izz                 =  2.0000
   Warping Constant    = 0.23299
   Torsion Constant    =  4.7330
   Center of Gravity Y  =-0.30973E-16
   Center of Gravity Z  = 0.15376E-15
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 Shear Center Y       =-0.22957E-13
 Shear Center Z       = 0.31281E-13

 Beam Section is offset to CENTROID of cross section
```

Menu Paths

Main Menu>Preprocessor>Sections>List Sections

SLOAD, *SECID*, *PLNLAB*, *KINIT*, *KFD*, *FDVALUE*, *LSLOAD*, *LSLOCK*

Loads a pretension section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SECID

Unique section number. The number must already be assigned to a pretension section.

PLNLAB

Label representing the pretension load sequence number in the format "PL*nn*" where *nn* is an integer from 1 through 99 (for example, PL01 through PL99).

Specify a value of DELETE to delete all loads on the specified pretension section (*SECID*). In this case, the command ignores any other argument values.

KINIT

Initial action key for pretension load PL01. (This field is omitted for PL02 and up.) Three scenarios are possible:

LOCK

Constrains (connects) the cutting plane on the pretension section. This value is the default.

SLID

Unconstrains (disconnects) the cutting plane on the pretension section.

TINY

Applies a very small pretension load (0.1% of *FDVALUE*) before the desired load is established. The small load prevents convergence problems which can occur when the desired load is not established in the first load step. This value is valid only if *KFD* = FORC.

KFD

Force/Displacement key. Specifies whether *FDVALUE* is a force or a displacement:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FORC

Apply a force on the specified pretension section. This value is the default.

DISP

Apply a displacement (adjustment) on the specified pretension section.

FDVALUE

Pretension load value. If *KFD* = FORC, this value is a pretension force. If *KFD* = DISP, this value is a pretension displacement (adjustment).

LSLOAD

Load step in which to apply the *FDVALUE*.

LSLOCK

The load step in which the displacement value resulting from the pretension force is locked. This value is valid only if *KFD* = FORC.

Command Default

The default pretension load value *FDVALUE* is zero (no load). A positive value puts the pretension elements in tension.

No default exists for the *LSLOAD* applied load step value. You must specify the load step in which to apply the *FDVALUE*.

No default exists for the *LSLOCK* locked load step value. You must specify the load step in which to lock the *FDVALUE*.

Notes

The **SLOAD (p. 1797)** command applies pretension loads to specified pretension sections (PRETS179-based) created via the **PSMESH (p. 1543)** command. A pretension load is ramp-applied (**KBC (p. 970)** = 0) if it is a force (*KFD* = FORC), and step-applied (**KBC (p. 970)** = 1) if it is a displacement (*KFD* = DISP).

You can lock the load value at a specified load step. When locked, the load changes from a force to a displacement, and the program applies the load as a constant displacement in all future load steps. Locking is useful when applying additional loadings. The additional loadings alter the effect of the initial load value, but because locking transforms the load into a displacement, it preserves the initial load's effect.

In modal and harmonic analyses, any pretension load (force, displacement, or locked) is ignored and no load is produced.

The **SLOAD (p. 1797)** command is not valid for MPC184-based preload sections created with **PSMESH (p. 1543)**.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example 13: Applying a Load

The following command shows how to establish loads on a pretension section:

```
SLOAD,1,PL01,TINY,FORC,5000,2,3
```

In this example, the load is applied to pretension section **1**, and the sequence begins with the initial action key, *KINIT*, set to TINY. A small stabilization load (5 = 0.10% of 5000) is applied in the first load step, as the actual pretension force is not applied until the second load step. The next four fields set the actual load: the *KFD* value FORC specifies the type of load, *FDVALUE* defines the pretension load value (5000), *LSLOAD* specifies the load step in which the force is applied (2), and the *LSLOCK* field specifies the load step in which the force is locked (3). Additional sets of four fields can be used to define additional loads.

Example 14: Editing an Existing Load

You can use the **SLOAD (p. 1797)** command to edit (overwrite) existing loads on a pretension section. This example changes the load on pretension section 1 (set above) to 6000:

```
SLOAD,1,PL01,,,6000,2,3
```

Unspecified values (blank fields), as shown in this example, remain unchanged from prior settings. If no prior specifications exist, then default values (*KINIT* = LOCK and *KFD* = FORC) apply.

Example 15: Deleting All Loads

The command can also delete all loads on a specified pretension section, as shown here:

```
SLOAD,1,DELETE
```

Example 16: Locking a Pretension Element

For a prestressed modal analysis, this command locks the pretension element:

```
SLOAD,1,PL01,LOCK,DISP,0,1,2
```

Multiple Loadings

The **SLOAD (p. 1797)** command allows you to apply multiple loadings. You can add up to 15 loadings (PL01 through PL15), or delete loadings, for any given pretension section(s).

Example 17: Applying Multiple Loadings

The following **SLOAD (p. 1797)** commands, issued in the order shown, establish a pretension load sequence in pretension section 2 with a force of 25 in load step (LS) 2, locked in LS 3-6, a force of 50 in LS 7, locked in LS 8-11, a force of 75 in LS 12, locked in LS 13 and beyond:

```
SLOAD,2,PL01,LOCK,FORC,25,2,3

SLOAD,2,PL02,,FORC,50,7,8

SLOAD,2,PL03,,FORC,75,12,13
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

At the same time, you can issue SLOAD commands to apply loads on *other* pretension sections. For example, in addition to the commands listed above, you could issue the following command to apply a load on pretension section 3:

```
SLOAD,3,PL01,LOCK,FORC,25,3,4
```

Using the Pretension Section Loads GUI

Any addition or deletion of a loading applies to the selected sections only. Mechanical APDL does not apply or delete a load until you click the **Apply** or **OK** button.

After you have successfully solved for a specified *LSLOAD* (GUI field **Apply at LS**) and eventually *LSLOCK* (GUI field **Lock at LS**) value, you cannot modify that loading's settings during subsequent steps of the analysis. Similarly, you cannot delete loadings that you have already partially or completely solved.

You can select more than one pretension section at a time in order to specify identical loadings on them. Before you completely solve a given loading, any combination of pretension sections is valid. The following limitations apply:

- After you have completely solved one or more loadings, Mechanical APDL allows multiple selection of *only* those pretension sections having

  -- the same number of defined loadings, *and*

  -- the identical loading number from the most recent completely solved loading.

- A multiple selection meeting the necessary criteria retains the settings that are identical for all selected pretension sections and leaves all other fields blank.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Pretnsn Sectn

Main Menu>Preprocessor>Loads>Define Loads>Delete>All Load Data>All Section Lds

Main Menu>Solution>Define Loads>Apply>Structural>Pretnsn Sectn

Main Menu>Solution>Define Loads>Delete>All Load Data>All Section Lds

---

SMALL, *IR*, *IA*, *IB*, *IC*, *Name*, - - , - - , *FACTA*, *FACTB*, *FACTC*

Finds the smallest of three variables.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IA*, *IB*, *IC

Reference numbers of the three variables to be operated on. If only two, leave *IC* blank. If only one, leave *IB* blank also.

Name

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

--, --

Unused fields.

FACTA*, *FACTB*, *FACTC

Scaling factors (positive or negative) applied to the corresponding variables (defaults to 1.0).

Notes

Finds the smallest of three variables according to the operation:

IR = smallest of (*FACTA* x *IA*, *FACTB* x *IB*, *FACTC* x *IC*)

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Find Minimum

***SMAT**, *Matrix*, *Type*, *Method*, *Val1*, *Val2*, *Val3*, *Val4* , *Val5*

Creates a sparse matrix.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Matrix

Name used to identify the matrix. Must be specified.

Type

Matrix type:

D --

Double precision real values (default).

Z --

Complex double precision values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method

Method used to create the matrix:

ALLOC --

Allocate a new matrix.

COPY --

Copy an existing matrix.

IMPORT --

Import the matrix from a file.

Val1, *Val2*, *Val3*, *Val4*, *Val5*

Additional input. The meaning of *Val1* through *Val5* will vary depending on the specified *Method*. See details below.

The following *Valx* fields are used with *Method* = ALLOC.

Val1

Method used to create the matrix:

DIAG --

Allocate a diagonal square matrix. *Val2* is used; *Val3*, *Val4*, and *Val5* are ignored.

Val2

Matrix size.

CSR --

Create a square sparse matrix based on Compressed Sparse Row (CSR) format description vectors. This format requires 3 input vectors specified as *Val2*, *Val3* and *Val4*.

Val2, *Val3*, *Val4*

Names of required *row_ptr*, *col_ind* and *val* vectors. These vectors must be created using the ***VEC (p. 2085)** command.

> **Note:**
>
> *row_ptr* is a long integer vector (use of L instead of I as the scalar type in the ***VEC (p. 2085)** call); *col_ind* is an integer vector. *Val* can be a real of complex values vector, according to the matrix type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Val5

Specifies whether the matrix is symmetric (TRUE) or unsymmetric (FALSE). Default = TRUE.

The following *Valx* fields are used with *Method* = COPY.

Val1

Name of the matrix to copy (can be either a dense or a sparse matrix).

Val2

Method used for copying the matrix:

DIAG --

Copy only the diagonal of the matrix. *Val3* and *Val4* are ignored.

TRANS --

Transpose the original matrix. *Val3* and *Val4* are ignored.

EXTRACT --

Extract a submatrix based on row and column numbers specified by *Val3* and *Val4*.

Val3

Name of integer vector (***VEC (p. 2085)**) containing row numbers. If no vector is specified, defaults to all rows.

Val4

Name of integer vector (***VEC (p. 2085)**) containing column numbers. If no vector is specified, defaults to all columns.

The following table describes the *Valx* fields used with *Method* = IMPORT.

***Method* = IMPORT**					
Val1	***Val2***	***Val3***	***Val4***	***Val5***	**Description**
FULL	File name [a]	Matrix type: STIFF - Stiffness matrix MASS - Mass matrix DAMP - Damping matrix NOD2SOLV - Mapping vector	(not used)	(not used)	Import a matrix from an existing FULL file. For a complex stiffness matrix, you can use the K_RE or K_IM label to import only the real part or only the imaginary part. The K_RE and

		relating the full set of nodal DOFs to the subset that the solver uses USR2SOLV - Mapping vector relating the full set of external nodal DOFs to the subset that the solver uses GMAT - Constraint equation matrix K_RE - Real part of the stiffness matrix K_IM - Imaginary part of the stiffness matrix			K_IM options both create real matrices. Note that the K_IM matrix is only available for **MODOPT (p. 1140)**,UNSYM and **MODOPT (p. 1140)**,DAMP modal analyses.
HBMAT	File name [a]	File format: ASCII BINARY	(not used)	(not used)	Import a matrix from an existing Harwell-Boeing format file.
MMF	File name [a]	(not used)	(not used)	(not used)	Import a matrix from an existing Matrix Market format file.
DMIG	File name [a]	File Format: FREE - Free field format LARGE - Large field format (default)	Delimeter character (only for FREE field format); defautl is ‘,’	Matrix name (default is the first matrix of the DMIG file)	Import a matrix from an existing NASTRAN `DMIG` file.
MAT	File name [a]	(not used)	(not used)	(not used)	Restore from a previous ***EXPORT (p. 716)** (*FORMAT* = MAT) command..

[a] File name is case-sensitive with a 32-character maximum.

Notes

Use the ***DMAT (p. 551)** command to create a dense matrix.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information on the CSR format, see Creating a Sparse Matrix Using the CSR Format in the *Ansys Parametric Design Language Guide*.

For more information on the NOD2SOLV and USR2SOLV mapping vectors, see Degree of Freedom Ordering in the *Ansys Parametric Design Language Guide*.

For more information about `.full` file contents, see the **HBMAT (p. 888)** in the *Command Reference* (p. 1).

Menu Paths

This command cannot be accessed from a menu.

SMAX, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*

Forms an element table item from the maximum of two other items.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1

First labeled result item in operation.

Lab2

Second labeled result item in operation (may be blank).

FACT1

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

Notes

Forms a labeled result item (see **ETABLE (p. 687)** command) for the selected elements by comparing two existing labeled result items according to the operation:

LabR = (*FACT1* x *Lab1*) cmx (*FACT2* x *Lab2*)

where "cmx" means "compare and save maximum." If absolute values are requested (**SABS (p. 1643)**,1), the absolute values of *Lab1* and *Lab2* are used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>Element Table>Find Maximum

/SMBC, *Mode*

Controls the display of solid model boundary condition symbols and labels.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Mode

CENT

Solid model boundary condition symbols and labels appear at the centroid of the solid model entity (default).

TESS

Solid model boundary condition symbols and labels appear inside each constituent element of the tessellation.

Notes

Mode = CENT is designed to reduce the clutter of boundary condition symbols in solid model plots. For example, if you have assigned normal pressure loads to an area, you may choose to display the pressures as arrows with the **/PSF (p. 1537)** command using **/PSF (p. 1537)**,PRES,NORM,2. When *Mode* = CENT, the pressure arrow is displayed at the centroid of the area. When *Mode* = TESS, a pressure arrow is displayed at the centroid of each polygon of the area's tessellation.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

SMBODY

Specifies "Body loads on the solid model" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu>**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List> Status. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Body Loads

SMCONS

Specifies "Constraints on the solid model" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>DOF Constraints

SMFOR

Specifies "Forces on the solid model" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Forces

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SMIN, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*

Forms an element table item from the minimum of two other items.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1

First labeled result item in operation.

Lab2

Second labeled result item in operation (may be blank).

FACT1

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

Notes

Forms a labeled result item (see **ETABLE (p. 687)** command) for the selected elements by comparing two existing labeled result items according to the operation:

LabR = (*FACT1* x *Lab1*) cmn (*FACT2* x *Lab2*)

where "cmn" means "compare and save minimum." If absolute values are requested (**SABS (p. 1643)**,1), the absolute values of *Lab1* and *Lab2* are used.

Menu Paths

Main Menu>General Postproc>Element Table>Find Minimum

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SMOOTH,*Vect1*,*Vect2*,*DATAP*,*FITPT*,*Vect3*,*Vect4*,*DISP*

Allows smoothing of noisy data and provides a graphical representation of the data.

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Vect1

Name of the first vector that contains the noisy data set (that is, independent variable). You must create and fill this vector before issuing **SMOOTH (p. 1809)**.

Vect2

Name of the second vector that contains the dependent set of data. Must be the same length as the first vector. You must create and fill this vector before issuing **SMOOTH (p. 1809)**.

DATAP

Number of data points to be fitted, starting from the beginning of the vector. If left blank, the entire vector will be fitted. The maximum number of data points is 100,000 (or greater, depending on the memory of the computer).

FITPT

Order of the fitting curve that will be used as a smooth representation of the data. This number should be less than or equal to the number of the data points. Default (blank) is one-half the number of data points. Maximum number of smoothed data fitting order is the number of data points up to 50. Depending on this number, the smoothed curve will be one of the following:

1

Curve is the absolute average of all of the data points.

2

Curve is the least square average of all of the data points.

3 or more

Curve is a polynomial of the order (n-1), where n is the number of data fitting order points.

Vect3

Name of the vector that contains the smoothed data of the independent variable. This vector should have a length equal to or greater than the number of smoothed data points. In batch (command) mode, you must create this vector before issuing the **SMOOTH (p. 1809)** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth_ind.

Vect4

Name of the vector that contains the smoothed data of the dependent variable. This vector must be the same length as *Vect3*. In batch (command) mode, you must create this vector before issuing

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the **SMOOTH (p. 1809)** command. In interactive mode, the GUI automatically creates this vector (if it does not exist). If you do not specify a vector name, the GUI will name the vector smth_dep.

DISP

Specifies how you want to display data. No default; you must specify an option.

1

Unsmoothed data only

2

Smoothed data only

3

Both smoothed and unsmoothed data

Notes

This command enables you to control the attributes of the graph using standard Mechanical APDL controls (**/GRID (p. 869)**, **/GTHK (p. 882)**, **/COLOR (p. 400)**, etc.).

If working interactively, the controls appear in this dialog box for convenience, as well as in their standard dialog boxes.

You must always create *Vect1* and *Vect2* (using ***DIM (p. 530)**) and fill these vectors before smoothing the data. If working interactively, the program automatically creates *Vect3* and *Vect4*. If working in batch (command) mode, you must create *Vect3* and *Vect4* (using ***DIM (p. 530)**) before issuing **SMOOTH (p. 1809)**. *Vect3* and *Vect4* are then filled automatically by the program.

The program also creates an additional TABLE type array that contains the smoothed array and the unsmoothed data to enable plotting later with ***VPLOT (p. 2146)**. Column 1 in the table corresponds to *Vect1*, column 2 to *Vect2*, and column 3 to *Vect4*. The array is named *Vect3*_SMOOTH, up to a limit of 32 characters. For example, if the array name is X1, the table name is X1_SMOOTH.

This command is also valid in PREP7 and SOLUTION.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SMRTSIZE, *SIZLVL*, *FAC*, *EXPND*, *TRANS*, *ANGL*, *ANGH*, *GRATIO*, *SMHLC*, *SMANC*, *MXITR*, *SPRX*

Specifies meshing parameters for automatic (smart) element sizing.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIZLVL

Overall element size level for meshing. The level value controls the fineness of the mesh. (Any input in this field causes remaining arguments to be ignored.) Valid inputs are:

n

Activate SmartSizing and set the size level to *n*. Must be an integer value from 1 (fine mesh) to 10 (coarse mesh). Remaining arguments are ignored, and argument values are set as shown in Table 226: SMRTSIZE - Argument Values for h-elements (p. 1813).

STAT

List current **SMRTSIZE (p. 1811)** settings.

DEFA

Set all **SMRTSIZE (p. 1811)** settings to default values (as shown in Table 226: SMRTSIZE - Argument Values for h-elements (p. 1813) for size level 6).

OFF

Deactivate SmartSizing. Current settings of **DESIZE (p. 519)** will be used. To reactivate SmartSizing, issue **SMRTSIZE (p. 1811)**,*n*.

FAC

Scaling factor applied to the computed default mesh sizing. Defaults to 1 for h-elements (size level 6), which is medium. Values from 0.2 to 5.0 are allowed.

EXPND

Mesh expansion (or contraction) factor. (This factor is the same as **MOPT (p. 1151)**,EXPND,*Value*.) *EXPND* is used to size internal elements in an area based on the size of the elements on the area's boundaries. For example, issuing **SMRTSIZE (p. 1811)**,,,2 before meshing an area will allow a mesh with elements that are approximately twice as large in the interior of an area as they are on the boundary. If *EXPND* is less than 1, a mesh with smaller elements on the interior of the area will be allowed. *EXPND* should be greater than 0.5 but less than 4. *EXPND* defaults to 1 for h-elements (size level 6), which does not allow expansion or contraction of internal element sizes (except when using **AESIZE (p. 104)** element sizing). (If *EXPND* is set to zero, the default value of 1 will be used.) The actual size of the internal elements will also depend on the *TRANS* option or upon **AESIZE (p. 104)** or **ESIZE (p. 668)** sizing, if used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TRANS

Mesh transition factor. (This factor is the same as **MOPT (p. 1151)**,TRANS,*Value*.) *TRANS* is used to control how rapidly elements are permitted to change in size from the boundary to the interior of an area. *TRANS* defaults to 2.0 for h-elements (size level 6), which permits elements to approximately double in size from one element to the next as they approach the interior of the area. (If *TRANS* is set to zero, the default value will be used.) *TRANS* must be greater than 1 and, for best results, should be less than 4. The actual size of the internal elements will also depend on the *EXPND* option or upon **AESIZE (p. 104)** or **ESIZE (p. 668)** sizing, if used.

ANGL

Maximum spanned angle per lower-order element for curved lines. Defaults to 22.5 degrees per element (size level 6). This angle limit may be exceeded if the mesher encounters a small feature (hole, fillet, etc.). (This value is *not* the same as that set by **DESIZE (p. 519)**,,,,*ANGL*.)

ANGH

Maximum spanned angle per higher-order element for curved lines. Defaults to 30 degrees per element (size level 6). This angle limit may be exceeded if the mesher encounters a small feature (hole, fillet, etc.). (This value is NOT the same as that set by **DESIZE (p. 519)**,,,,,*ANGH*.)

GRATIO

Allowable growth ratio used for proximity checking. Defaults to 1.5 for h-elements (size level 6). Values from 1.2 to 5.0 are allowed; however, values from 1.5 to 2.0 are recommended.

SMHLC

Small hole coarsening key, can be ON (default for size level 6) or OFF. If ON, this feature suppresses curvature refinement that would result in very small element edges (that is, refinement around small features).

SMANC

Small angle coarsening key, can be ON (default for all levels) or OFF. If ON, this feature restricts proximity refinement in areas where it is ill-advised (that is, in tight corners on areas, especially those that approach 0 degrees).

MXITR

Maximum number of sizing iterations (defaults to 4 for all levels).

SPRX

Surface proximity refinement key, can be off (*SPRX* = 0, *which is the default for all levels*) or on via two different values (*SPRX* = 1 or *SPRX* = 2). If *SPRX* = 1, surface proximity refinement is performed and any shell elements that need to be modified are modified. If *SPRX*=2, surface proximity refinement is performed but no shell elements are altered.

Command Default

SmartSizing is off and **DESIZE (p. 519)** is used for automatic element sizing.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

If a valid level number (1 (fine) to 10 (coarse)) is input on *SIZLVL*, inputs for remaining arguments are ignored, and the argument values are set as shown in Table 226: SMRTSIZE - Argument Values for h-elements (p. 1813).

The settings shown are for h-elements. The first column contains *SIZLV* data, ranging from 10 (coarse) to 1 (fine). The default is 6 (indicated by the shaded row).

Table 226: SMRTSIZE - Argument Values for h-elements

	FAC	EXPND	TRANS	ANGL	ANGH	GRATIO	SMHLC	SMANC	MXITR	SPRX
10	5.0	2.0	2.0*	45.0	45.0*	2.0	on	on	4*	off
9	3.0	1.75	2.0*	36.0	45.0*	1.9	on	on	4*	off
8	1.875	1.5	2.0*	30.0	45.0*	1.8	on	on	4*	off
7	1.5	1.0	2.0*	26.0	36.0*	1.7	on	on	4*	off
6	1.0*	1.0*	2.0*	22.5	30.0*	1.5*	on	on	4*	off
5	0.65	1.0*	2.0*	18.0	27.0	1.5	on	on	4*	off
4	0.4	1.0*	2.0*	15.0	22.0	1.5	off	on	4*	off
3	0.3	1.0*	2.0*	12.0	18.0	1.5	off	on	4*	off
2	0.25	1.0*	2.0*	10.0	15.0	1.5	off	on	4*	off
1	0.2	1.0*	2.0*	7.5	15.0	1.4	off	on	4*	off

Where appropriate, SmartSizing will start with **AESIZE (p. 104)** settings. Elsewhere, it will start with any defined **ESIZE (p. 668)**,SIZE setting. It will locally override **AESIZE (p. 104)** or **ESIZE (p. 668)** for proximity and curvature. SmartSizing ignores any **ESIZE (p. 668)**,,NDIV setting.

LESIZE (p. 1041) line division and spacing specifications will be honored by SmartSizing, unless you give permission for SmartSizing to override them (for proximity or curvature) by setting KYNDIV to 1. Lines not having an **LESIZE (p. 1041)** specification are meshed as well as they can be.

This command is also valid for rezoning.

Menu Paths

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Basic

Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Status

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SMSURF

Specifies "Surface loads on the solid model" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>Surface Loads

SMULT, *LabR*, *Lab1*, *Lab2*, *FACT1*, *FACT2*

Forms an element table item by multiplying two other items.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to results. If same as existing label, the existing values will be overwritten by these results.

Lab1

First labeled result item in operation.

Lab2

Second labeled result item in operation (may be blank).

FACT1

Scale factor applied to *Lab1*. A (blank) or '0' entry defaults to 1.0.

FACT2

Scale factor applied to *Lab2*. A (blank) or '0' entry defaults to 1.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Forms a labeled result item (see **ETABLE (p. 687)** command) for the selected elements by multiplying two existing labeled result items according to the operation:

LabR = (*FACT1* x *Lab1*) x (*FACT2* x *Lab2*)

May also be used to scale results for a single labeled result item. If absolute values are requested (**SABS (p. 1643)**,1), the absolute values of *Lab1* and *Lab2* are used.

Menu Paths

Main Menu>General Postproc>Element Table>Multiply

SNOPTION,*RangeFact*,*BlockSize*,*RobustLev*,*Compute*,--,*Solve_Info*

Specifies Supernode (SNODE) eigensolver options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RangeFact

Factor used to control the range of eigenvalues computed for each supernode. The value of *RangeFact* must be a number between 1.0 and 10.0. By default the *RangeFact* value is set to 2.0, which means that all eigenvalues between 0 and 2**FREQE* are computed for each supernode (where *FREQE* is the upper end of the frequency range of interest as specified on the **MODOPT (p. 1140)** command). As the *RangeFact* value increases, the eigensolution for the SNODE solver becomes more accurate and the computational time increases.

BlockSize

BlockSize to be used when computing the final eigenvectors. The value of *Blocksize* must be either MAX or a number between 1 and *NMODE*, where *NMODE* is the number of modes to be computed as set on the **MODOPT (p. 1140)** command. Input a value of MAX to force the algorithm to allocate enough memory to hold all of the final eigenvectors in memory and, therefore, only read through the file containing the supernode eigenvectors once. Note that this setting is ONLY recommended when there is sufficient physical memory on the machine to safely hold all of the final eigenvectors in memory.

RobustLev

Parameter used to control the robustness of the SNODE eigensolver. The value of *RobustLev* must be a number between 0 and 10. Lower values of *RobustLev* allow the eigensolver to run in the most efficient manner for optimal performance. Higher values of *RobustLev* often slow down the performance of the eigensolver, but can increase the robustness; this may be desirable if a problem is detected with the eigensolver or its eigensolution.

Compute

Key to control which computations are performed by the Supernode eigensolver:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EVALUE

The eigensolver computes only the eigenvalues.

EVECTOR

The eigensolver computes only the eigenvectors (must be preceded by a modal analysis where the eigenvalues were computed using the Supernode eigensolver).

BOTH

The eigensolver computes both the eigenvalues and eigenvectors in the same pass (default).

--

Unused field

Solve_Info

Solver output option:

OFF

Turns off additional output printing from the Supernode eigensolver (default).

PERFORMANCE

Turns on additional output printing from the Supernode eigensolver, including a performance summary and a summary of file I/O for the Supernode eigensolver. Information on memory usage during assembly of the global matrices (that is, creation of the `Jobname.full` file) is also printed with this option.

Command Default

RangeFact = 2.0. *BlockSize* is set to min(*NMODE*,40), where *NMODE* is the number of modes to be computed as set on the **MODOPT (p. 1140)** command. *RobustLev* = 0. *Compute* = BOTH. Additional output is not printed (*Solve_Info* = OFF).

Notes

This command specifies options for the Supernode (SNODE) eigensolver.

Setting *RangeFact* to a value between 2.0 and 10.0 will improve the accuracy of the computed eigenvalues and eigenvectors, but will often increase the computing time of the SNODE eigensolver. Conversely, setting *RangeFact* to a value less than 2.0 will deteriorate the accuracy of the computed eigenvalues and eigenvectors, but will often speed up the computing time of the SNODE eigensolver. The default value of 2.0 has been set as a good blend of accuracy and performance. If the model has rigid body modes, setting *RangeFact* higher than 2 is recommended to achieve better solution accuracy for the lower flexible modes.

The SNODE eigensolver reads the eigenvectors and related information for each supernode from a file and uses that information to compute the final eigenvectors. For each eigenvalue/eigenvector requested by the user, the program must do one pass through the entire file that contains the supernode eigen-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

vectors. By choosing a *BlockSize* value greater than 1, the program can compute *BlockSize* number of final eigenvectors for each pass through the file. Therefore, smaller values of *BlockSize* result in more I/O, and larger values of *BlockSize* result in less I/O. Larger values of *BlockSize* also result in significant additional memory usage, as *BlockSize* number of final eigenvectors must be stored in memory. The default *Blocksize* of min(*NMODE*,40) is normally a good choice to balance memory and I/O usage.

The *RobustLev* field should only be used when a problem is detected with the accuracy of the final solution or if the Supernode eigensolver fails while computing the eigenvalues/eigenvectors. Setting *RobustLev* to a value greater than 0 will cause the performance of the eigensolver to deteriorate. If the performance deteriorates too much or if the eigensolver continues to fail when setting the *RobustLev* field to higher values, then switching to another eigensolver such as Block Lanczos or PCG Lanczos is recommended.

Setting *Compute* = EVALUE causes the Supernode eigensolver to compute only the requested eigenvalues. During this process a Jobname.SNODE file is written; however, a Jobname.mode file is not written. Thus, errors will likely occur in any downstream computations that require the Jobname.mode file (for example, participation factor computations, mode superpostion transient/harmonic analysis, PSD analysis). Setting *Compute* = EVECTOR causes the Supernode eigensolver to compute only the corresponding eigenvectors. The Jobname.SNODE file and the associated Jobname.full file are required when requesting these eigenvectors. In other words, the eigenvalues must have already been computed for this model before computing the eigenvectors. This field can be useful in order to separate the two steps (computing eigenvalues and computing eigenvectors).

For more information on the eigensolver's accuracy and a discussion of its known limitations, see Supernode Method in the *Mechanical APDL Theory Reference*

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

SOLOPTION, *Option*, *Type*, *Value*

Specifies solution transition options.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Transition option:

STOT

Use criterion for transitioning from a static solution to a transient dynamic solution.

TTOS

Use criterion for transitioning from a transient dynamic solution to a static solution.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TTOT

Use criterion for transitioning from one time integration algorithm to another for a transient dynamic solution between load steps.

Type

Additional input that varies depending on the *Option* value. See table below.

Value

Additional input that varies depending on the *Option* and *Type* values. See table below.

Option Label	*Type* =	*Value* =
STOT (default)	CONV – Transition from a static solution to a transient solution when the static solution fails to converge (default).	Time integration scheme for the transient solution: QUASI - Quasi static transient (default) NMK - Newmark time integration HHT - HHT time integration
	FORC – Transition from a static solution to a transient solution upon reading this command (useful in forcing a transition during a restart).	
	RBM – Transition from a static solution to a transient solution the first time the solution experiences rigid body motion, which is determined by maximum DOF increment. The transition occurs if the maximum DOF increment is more than the criterion specified (or the default) on the **CUTCONTROL (p. 445)** command.	
	CONT – Transition from a static solution to a transient solution the first time any contact pair goes from having some points in contact to the entire pair being in a far-field status.	
TTOS	TIME – Time to spend in the transient solution before an attempt is made to come back into a static solution.	Time value
TTOT	FORC – Transition from one time integration algorithm to another upon reading this command. This option can be used between load steps or during restart.	Time integration scheme for the transient solution: QUASI - Quasi static transient (default)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option Label	*Type* =	*Value* =
		NMK - Newmark time integration HHT - HHT time integration

Notes

This command triggers an automatic transition from a static solution to a transient solution based on the specified criterion. The command is valid only in the solution processor (**/SOLU (p. 1821)**) and must be defined either before the first **SOLVE (p. 1822)** command or during a restart analysis.

If **SOLOPTION (p. 1817)** is issued with no arguments specified, the static solution will transition to a quasi-static transient solution if the static solution fails to converge (that is, **SOLOPTION (p. 1817)**,STOT,CONV,QUASI).

SOLOPTION (p. 1817) can be used in a restart even if the base analysis did not include the command. Therefore, a problem that failed in the static analysis can be restarted using this command so that it transitions to a transient solution and solves further. Material densities are required for the transient solution, but they must be defined during the static solution since the restart framework does not permit material density to be defined in the restart analysis.

For more information on using **SOLOPTION (p. 1817)**, see Automatic Transition Between Static and Transient Solutions in the *Advanced Analysis Guide*.

Menu Paths

This command cannot be accessed from a menu.

SOLU,*NVAR*,*Item*,*Comp*,*Name*

Specifies solution summary data per substep to be stored.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

Arbitrary reference number assigned to this variable (2 to *NV* (**NUMVAR (p. 1302)**)).

Item

Label identifying the item. Valid item labels are shown in the table below. Some items may also require a component label.

Comp

Component of the item (if required). Valid component labels are shown in the table below. None are currently required.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

Thirty-two character name identifying the item on printouts and displays. Defaults to an eight character label formed by concatenating the first four characters of the *Item* and *Comp* labels.

Notes

See also the **PRITER (p. 1497)** command of POST1 to display some of these items directly. Valid for a static or full transient analysis. All other analyses have zeros for the data. Valid item and component labels for solution summary values are:

Valid Item Labels

Item	Description
ALLF	Total arc-length load factor (ratio of the load at equilibrium to the total applied load)
ALDLF	Arc-length load factor increment (change in ALLF)
ARCL	Normalized arc-length radius
CNVG	Convergence indicator.
CRPRAT	Maximum creep ratio.
CSCV	Current segment convergence value.
CUCV	Current convergence value.
DICV	Displacement convergence value.
DSPRM	Descent parameter.
DTIME	Time step size.
EQIT	Number of equilibrium iterations.
FFCV	Fluid flow convergence value.
FOCV	Force convergence value.
HFCV	Heat flow convergence value.
NCMIT	Cumulative number of iterations.
NCMLS	Cumulative number of load steps
NCMSS	Cumulative number of substeps
MFCV	Magnetic flux convergence value.
MOCV	Moment convergence value.
MXDVL	Maximum degree of freedom value.
PRCV	Pressure convergence value.
PSINC	Maximum plastic strain increment.
RESFRQ	Response frequency for 2nd order systems.
RESEIG	Response eigenvalue for 1st order systems.
ROCV	Rotation convergence value.
SMCV	Scalar magnetic potential convergence value.
TECV	Temperature convergence value.
VECV	Velocity convergence value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valid Item Labels

Item	Description
VOCV	Voltage convergence value.
VMCV	Vector magnetic potential convergence value.

Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

/SOLU

Enters the solution processor.

SESSION (p. 11): Processor Entry (p. 12)
SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command is valid only at the Begin Level.

Menu Paths

Main Menu>Solution

SOLUOPT

Specifies "Solution options" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Solution>General

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SOLVE,*Action*

Starts a solution.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | PrepPost | **Solver** | AS add-on

Action

Action to be performed on solve (used only for linear perturbation analyses).

ELFORM

Reform all appropriate element matrices in the first phase of a linear perturbation analysis.

Notes

Starts the solution of one load step of a solution sequence based on the current analysis type and option settings. Use *Action* = ELFORM only in the first phase of a linear perturbation analysis.

Menu Paths

Main Menu>Solution>Solve>Current LS

***SORT**,*Name*,*SortType*,*Val1*,*Val2*

Sorts the values of the specified vector.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name

Name of the vector to be sorted. This vector can contain real or complex values.

SortType

Criteria used to sort the values:

VALUE –

Values are sorted based on their real value (default).

ABS –

Values are sorted based on their absolute value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PERM –

Values are sorted based on the input permutation vector (*Val1*).

Val1, Val2

Additional input. The meaning of *Val1*, *Val2* varies depending on the specified *SortType*. See below for details.

The following *Valx* fields are used with *SortType* = VALUE or ABS:

Val1

Name of the vector in which permutation values will be saved (optional). This should be an empty vector of type integer that was created with the ***VEC (p. 2085)** command. The size of this permutation vector must be identical to the size of the vector to be sorted. After the sort, this vector contains the permutation used to sort the values.

Val2

Order of the sort operation:

0 –

Increasing order (default).

1 –

Decreasing order.

The following *Valx* fields are used with *Method* = PERM:

Val1

Name of the permutation vector used to sort the values. This must be a vector of integer values that was created with the ***VEC (p. 2085)** command. The size of this permutation vector must be identical to the size of the vector to be sorted.

This permutation vector is required when using *Method* = PERM.

Notes

The examples below demonstrate using ***SORT (p. 1822)** to sort the values of an input vector.

The following input:

```
*VEC,V,I,ALLOC,5
V(1)=5,-3,2,0,-1
*SORT,V,VALUE
*PRINT,V
```

generates this output:

```
       -3        -1         0         2         5
```

To reverse the order, this input:

```
*SORT,V,VALUE,,1
*PRINT,V
```

generates this output:

```
        5        2        0       -1       -3
```

Menu Paths

This command cannot be accessed from a menu.

SORT

Specifies "Sort settings" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Sort Module

SOURCE, *X*, *Y*, *Z*

Defines a default location for undefined nodes or keypoints.

PREP7 (p. 22): Keypoints (p. 25)
PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X, Y, Z

Global Cartesian coordinates for source nodes or keypoints (defaults to the origin).

Command Default

Global Cartesian origin.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines a global Cartesian location for undefined nodes or keypoints moved during intersection calculations (**MOVE (p. 1158)** or **KMOVE (p. 986)**).

Menu Paths

This command cannot be accessed from a menu.

SPACE, *NODE*

Defines a space node for radiation using the Radiation Matrix method.

AUX12 (p. 62): Radiation Matrix Method (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node defined to be the space node.

Command Default

No space node (no radiation to space).

Notes

A space node is required in an open system to account for radiation losses.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Other Settings

SPCNOD, *ENCL*, *NODE*

Defines a space node for radiation using the Radiosity method.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ENCL

Radiating surface enclosure number. Defaults to 1. If ENCL = STAT, the command lists all enclosure space nodes. If ENCL = DELE, the command deletes all enclosure space nodes.

NODE

Node defined to be the space node.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

For open systems, an enclosure may radiate to a space node (*NODE*).

Open systems may be characterized by one or more enclosures (*ENCL*). Each enclosure may radiate to a different space node (*NODE*).

For a space node that is not part of the finite element model, specify the temperature using the **D (p. 483)** command. For the first load step, the space node temperature ramps from the uniform temperature specified by the **TUNIF (p. 2027)** command to the temperature specified by the **D (p. 483)** command. For subsequent load steps, it ramps from the previous value of the space node temperature. For intermediate load steps, use the **SPCNOD (p. 1825)**,DELETE command and specify the space node temperature again to ramp from the uniform temperature.

For a space node that is part of the finite element model, the temperature is that calculated during the finite element solution.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Solution Opt

Main Menu>Radiation Opt>Radiosity Meth>Solution Opt

Main Menu>Solution>Radiation Opts>Solution Opt

SPCTEMP, *ENCL*, *TEMP*

Defines a free-space ambient temperature for radiation using the Radiosity method.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ENCL

Radiating surface enclosure number. Defaults to 1. If *ENCL* = STAT, the command lists all enclosure space temperatures. If *ENCL* = DELE, the command deletes all enclosure space temperatures.

TEMP

Temperature of free-space in the reference temperature system. The temperature will be offset by the value specified in the **TOFFST (p. 2008)** command for internal calculations.

Notes

For open systems, an enclosure may radiate to the free-space ambient temperature (*TEMP*).

Open systems may be characterized by one or more enclosures (*ENCL*). Each enclosure may radiate to a different free-space ambient temperature (*TEMP*).

For the first load step, the space temperature ramps from the uniform temperature specified by the **TUNIF (p. 2027)** command to the temperature specified by the **SPCTEMP (p. 1826)** command. For subsequent

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

load steps, it ramps from the previous value of the space temperature. For intermediate load steps, use the **SPCTEMP (p. 1826)**,DELETE command and specify the space temperature again to ramp from the uniform temperature.

Important:

Reissuing **SPCTEMP (p. 1826)** does not overwrite the previous value. To change the free-space ambient temperature (*TEMP*) between loadsteps, you must issue **SPCTEMP (p. 1826)**,DELETE and then reissue **SPCTEMP (p. 1826)**,*ENCL*,*TEMP*.

Menu Paths

Main Menu>Preprocessor>Radiation Opts>Solution Opt

Main Menu>Radiation Opt>Radiosity Meth>Solution Opt

Main Menu>Solution>Radiation Opts>Solution Opt

SPDAMP, *TBLNO*, *CURVNO*, *DampRatio*

Defines input spectrum damping in a multi-point response spectrum analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Input table number. Corresponds to the frequency table number (*TBLNO* on the **SPFREQ (p. 1828)** command).

CURVNO

Input curve number. Corresponds to the spectrum values curve number (*CURVNO* on the **SPVAL (p. 1842)** command).

DampRatio

Damping ratio for the response spectrum curve. Up to 20 different curves may be defined, each with a different damping ratio. Damping values must be input in ascending order.

Notes

Defines multi-point response spectrum damping value to be associated with:

- Previously defined frequency points (**SPFREQ (p. 1828)**).
- Subsequently defined spectrum points (**SPVAL (p. 1842)**).

Damping values are used only to identify input spectrum values for the mode coefficients calculation.

The curve number must be input in ascending order starting with 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Spect vs Freq

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Spect vs Freq

---

SPEC

Specifies "Miscellaneous specifications" as the subsequent status topic.

POST1 (p. 51): Status (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>General Postproc>Output Options

SPFREQ, *TBLNO*, *FREQ1*, *FREQ2*, *FREQ3*, *FREQ4*, *FREQ5*, *FREQ6*, *FREQ7*

Defines the frequency points for the input spectrum tables SPVAL (p. 1842) vs. SPFREQ (p. 1828) for multi-point spectrum analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Input table number. Up to 200 tables may be defined.

FREQ1, FREQ2, FREQ3,..., FREQ7

Frequency points (Hz) for spectrum vs. frequency tables. *FREQ1* should be greater than zero, and values must be in ascending order.

Notes

The spectrum values are input with the **SPVAL (p. 1842)** command. A separate **SPFREQ (p. 1828)** command must be used for each table defined. Frequencies must be in ascending order.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Repeat **SPFREQ (p. 1828)** command for additional frequency points. Values are added after the last nonzero frequency.

If all fields after **SPFREQ (p. 1828)** are blank, all input vs. frequency tables are erased. If *TBLNO* is the only non-blank field, all corresponding **SPVAL (p. 1842)** curves are erased.

Use the **SPTOPT (p. 1840)** and **STAT (p. 1861)** commands to list current frequency points.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Erase Tables

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Freq points

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Erase Tables

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Freq points

---

SPGRAPH, *TBLNO*, *CURVNO*, *CURVNOBeg*

Displays input spectrum curves for MPRS analysis.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Table number to display. Defaults to 1.

CURVNO

Curve number to display. Defaults to none.

CURVNOBeg

Beginning of the curve number range to display. Defaults to 1.

Notes

You can display up to 10 input spectrum curves (**SPVAL (p. 1842)** and **SPFREQ (p. 1828)** commands) with log X scale.

If the input spectrum curves are not associated with a damping value (**SPDAMP (p. 1827)** command), *CURVNO* and *CURVNOBeg* are not applicable and table *TBLNO* is displayed. Otherwise, specify *CURVNO* or *CURVNOBeg*:

- if *CURVNO* is used, one curve is displayed.
- if *CURVNOBeg* is used, up to 10 curves are displayed. *CURVNOBeg* is the beginning of the curve number range of interest.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

SPH4, *XCENTER*, *YCENTER*, *RAD1*, *RAD2*

Creates a spherical volume anywhere on the working plane.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XCENTER, YCENTER

Working plane X and Y coordinates of the center of the sphere.

RAD1, RAD2

Inner and outer radii (either order) of the sphere. A value of zero or blank for either *RAD1* or *RAD2* defines a solid sphere.

Notes

Defines either a solid or hollow spherical volume anywhere on the working plane. The sphere must have a spatial volume greater than zero. (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) A sphere of 360° will be defined with two areas, each consisting of a hemisphere. See the **SPHERE (p. 1831)** and **SPH5 (p. 1830)** commands for other ways to create spheres.

When working with a model imported from an IGES file (DEFAULT import option), you can create only solid spheres. If you enter a value for both *RAD1* and *RAD2* the command is ignored.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Primitives>Solid Sphere

Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>Hollow Sphere

Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>Solid Sphere

---

SPH5, *XEDGE1*, *YEDGE1*, *XEDGE2*, *YEDGE2*

Creates a spherical volume by diameter end points.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XEDGE1, YEDGE1

Working plane X and Y coordinates of one edge of the sphere.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

XEDGE2, YEDGE2

Working plane X and Y coordinates of the other edge of the sphere.

Notes

Defines a solid spherical volume anywhere on the working plane by specifying diameter end points. The sphere must have a spatial volume greater than zero. (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) A sphere of 360° will be defined with two areas, each consisting of a hemisphere. See the **SPHERE (p. 1831)** and **SPH4 (p. 1830)** commands for other ways to create spheres.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>By End Points

---

SPHERE, *RAD1*, *RAD2*, *THETA1*, *THETA2*

Creates a spherical volume centered about the working plane origin.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RAD1, RAD2

Inner and outer radii (either order) of the sphere. A value of zero or blank for either *RAD1* or *RAD2* defines a solid sphere.

THETA1, THETA2

Starting and ending angles (either order) of the sphere. Used for creating a spherical sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0.0° and the ending angle defaults to 360.0°. See the *Modeling and Meshing Guide* for an illustration.

Notes

Defines either a solid or hollow sphere or spherical sector centered about the working plane origin. The sphere must have a spatial volume greater than zero. (that is, this volume primitive command cannot be used to create a degenerate volume as a means of creating an area.) Inaccuracies can develop when the size of the object you create is much smaller than the relative coordinate system values (ratios near to or greater than 1000). If you require an exceptionally small sphere, create a larger object, and scale it down to the appropriate size.

For a solid sphere of 360°, you define it with two areas, each consisting of a hemisphere. See the **SPH4 (p. 1830)** and **SPH5 (p. 1830)** commands for the other ways to create spheres.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Sphere>By Dimensions

---

SPLINE, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *XV1*, *YV1*, *ZV1*, *XV6*, *YV6*, *ZV6*

Generates a segmented spline through a series of keypoints.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1, P2, P3, . . . , P6

Keypoints through which the spline is fit. At least two must be defined. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

The following fields are used only if specified end slopes on the line are desired, otherwise zero curvature end slopes will be automatically calculated to produce a line which is "straight" in the active coordinate system. To specify end slopes, use the following fields to define a "slope vector" (one for each end of the line, if desired) that has its tail at the origin and its head at the point *XVn*,*YVn*, *ZVn* in the active coordinate system (**CSYS (p. 441)**). The corresponding end slope of the line will then be parallel to this "slope vector."

XV1, YV1, ZV1

Location (in the active coordinate system) of the head of the "slope vector" corresponding to the slope at the *P1* end of the spline. The tail of the vector is at the origin of the coordinate system.

XV6, YV6, ZV6

Location of the head of the "slope vector" corresponding to the slope at the *P6* (or the last keypoint if fewer than six specified) end of the spline.

Notes

The output from this command is a series of connected lines (one line between each pair of keypoints) that together form a spline. Note that solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>Segmented Spline

Main Menu>Preprocessor>Modeling>Create>Lines>Splines>With Options>Segmented Spline

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SPLOT, *NA1*, *NA2*, *NINC*, *MESH*

Displays the selected areas and a faceted view of their underlying surfaces

PREP7 (p. 22): Areas (p. 27)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1

Starting area for display of areas and underlying surfaces. If *NA1* = ALL (default), *NA2* and *NINC* are ignored and all selected areas are displayed (**ASEL (p. 192)** command).

NA2

Last area to be displayed.

NINC

Numeric value setting steps between NA1 and NA2 for display. Default value is (1).

MESH

Specifies a rectangular mesh density used to display the underlying surface (default 4, i.e. 4 x 4).

Notes

This command is valid in any processor. The plot output displays the external and internal trim curves and underlying surface. You cannot obtain a faceted view of your surface areas when you are using the **/EXPAND (p. 712)** command to create larger graphics displays.

Use **APLOT (p. 169)** for trimmed surface display.

Menu Paths

This command cannot be accessed from a menu.

SPMWRITE, *Method*, *NMODE*, *Inputs*, *InputLabels*, *Outputs*, *OutputLabels*, *NIC*, *VelAccKey*, *FileFormat*

Calculates the state-space matrices and writes them to the SPM file.

POST1 (p. 51): Special Purpose (p. 56)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Method

Reduction method for the calculation of the state-space matrices.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MODAL

Method based on modal analysis results from LANB, LANPCG, SNODE, or SUBSP eigensolver (default).

NMODE

Number of modes to be used. Defaults to all modes.

Inputs

Definition of the inputs. Defaults to all load vectors on the MODE file.

If an integer is entered, it specifies the number of load vectors from the MODE file used for the definition of the inputs. The first *Inputs* load vectors are used.

If *Inputs* is an array parameter, the first column is the node number and the second column is the structural degree of freedom (1=UX, 2=UY, 3=UZ, 4=ROTX, 5=ROTY, 6=ROTZ) indicating input points. The number of rows in the array parameter is equal to the number of inputs.

InputLabels

Definition of the input labels. Defaults to the load vector numbers or input definition (node and degree of freedom array parameter), depending on the *Inputs* specification.

If a character array parameter is entered (Type=CHAR in the ***DIM (p. 530)** command), each 8 character string represents an input label. Only valid when *Inputs* is an array parameter

Outputs

Definition of the outputs. Defaults to the inputs.

If an array parameter is entered, the first column is the node number and the second column is the structural degree of freedom (1=UX, 2=UY, 3=UZ, 4=ROTX, 5=ROTY, 6=ROTZ) of the output points. The number of rows in the array parameter is equal to the number of outputs.

OutputLabels

Definition of the output labels. Defaults to the output definition (node and degree of freedom) if used, else defaults to the *InputLabels*.

If a character array parameter is entered (Type=CHAR in the *DIM command), each 8 character string represents an output label.

NIC

Load vector on the *MODE* file used for the calculation of the initial conditions. Defaults to no initial condition.

VelAccKey

Output velocities and accelerations key.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Output displacements only (default).

ON

Output displacements, velocities and accelerations.

FileFormat

The format of the SPM file.

0

Dense format.

1

Matrix Market Exchange format (non-zero terms only).

2

Twin Builder SML format without reference (default).

3

Twin Builder SML format with common reference.

4

Twin Builder SML format with independent references.

Notes

The SPMWRITE generates the file `Jobname.spm` containing the state-space matrices and other information.

The following applies to the SML formats (FileFormat = 2, 3, and 4):

- For conservative systems where the outputs are equal to the inputs (*`Outputs`* is left blank):
  - The labels for the inputs (*`InputLabels`*) are required.
  - The *`Inputs`* must use the array parameter option so that the input degrees of freedom (DOFs) are known.
- For non-conservative systems where the outputs are not equal to the inputs:
  - The labels for the outputs (*`OutputLabels`*) are required.
  - The file formats with references (*`FileFormat`* = 3 and 4) do not apply.
- Velocity and acceleration results are not included in the state-space matrices calculation (VelAccKey = OFF)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- File format with common reference (FileFormat = 3) does not apply if the inputs are based on DOFs of a different nature. All input DOFs must be either all rotational or all translational and not a mix of the two.
- A graphics file (`Jobname_SPM.png`) is generated. It contains an element plot of the model.

For more details about the reduction method and the generation of the state-space matrices, see Reduced-Order Modeling for State-Space Matrices Export in the *Mechanical APDL Theory Reference*.

For examples of the command usage, see State-Space Matrices Export.

Menu Paths

This command cannot be accessed from a menu.

SPOINT, *NODE*, *X*, *Y*, *Z*, *InertiaKey*

Defines a point for force/moment summations or inertia calculation

POST1 (p. 51): Special Purpose (p. 56)
SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NODE

Node number of the desired point. If zero, use *X*,*Y*,*Z* to describe point.

X*, *Y*, *Z

Global Cartesian coordinates of the desired summation point. Used if *NODE* is 0. Defaults to (0,0,0).

InertiaKey

Inertia key:

OFF

Point or node is used for the force/moment summations (default).

ON

Point or node is used for the calculation of total inertia.

Command Default

No point is defined by default; you must either specify a node or coordinates.

Notes

By default (*InertiaKey* = OFF), defines a point (any point other than the origin) about which the tabular moment summations are computed. If force summations are desired in other than the global

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian directions, a node number must be specified on the *NODE* field, *and* the desired coordinate system must be activated with **RSYS (p. 1639)**. The command must be issued in the **/POST1 (p. 1461)** module.

When the inertia key is activated (*InertiaKey* = ON), the total inertia printed in the precise mass summary is calculated with respect to the point or node in the global Cartesian system. In this case, the command must be issued during the first load step in the **/SOLU (p. 1821)** module.

Menu Paths

Main Menu>General Postproc>Nodal Calcs>Summation Pt>At Node

Main Menu>General Postproc>Nodal Calcs>Summation Pt>At XYZ Loc

---

SPOPT,*Spectype*,*NMODE*,*Elcalc*,*ModeReuseKey*,*SpecCum*

Selects the spectrum type and other spectrum options.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Spectype

Spectrum type:

SPRS

Single point excitation response spectrum (default). See also **SVTYP (p. 1887)**.

MPRS

Multiple point excitation response spectrum.

DDAM

Dynamic design analysis method.

PSD

Power spectral density.

NMODE

Use the first *NMODE* modes from the modal analysis. Defaults to all extracted modes, as specified by the **MODOPT (p. 1140)** and **BUCOPT (p. 280)** commands. *NMODE* cannot be larger than 10000.

Elcalc

Element results calculation key:

NO

Do not calculate element results and reaction forces (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

YES

Calculate element results and reaction forces, as well as the nodal degree of freedom solution.

ModeReuseKey

Key for existing *MODE* file reuse when running multiple spectrum analyses:

NO

No spectrum analysis has been performed yet (default).

YES

This is not the first spectrum analysis. The *MODE* file will be reused and the necessary files will be cleaned up for the new spectrum analysis.

SpecCum

Key to cumulate all response spectra solutions in `Jobname.rst` when multiple response spectra are calculated (only supported for SPRS and DDAM analyses with *Elcalc* = YES):

NO

Do not cumulate the response spectra solutions (default).

YES

Cumulate the response spectra solutions.

Notes

Valid only for a spectrum analysis (**ANTYPE (p. 162)**,SPECTR). This operation must be preceded by a modal solution (**ANTYPE (p. 162)**,MODAL) with the appropriate files available. Both the spectrum analysis and the preceding modal analysis must be performed under the same Mechanical APDL version number.

If used in SOLUTION, this command is valid only within the first load step.

Element results are calculated (*Elcalc* = YES) only if the element modal results are available (written to the `Jobname.mode` file with *MSUPkey* = YES on the **MXPAND (p. 1203)** command). For *Sptype* = SPRS, MPRS, and DDAM, if the element results calculation is activated (*Elcalc* = YES) and element modal results are not available, it is deactivated automatically.

For SPRS, DDAM or MPRS analyses, when *Elcalc* = YES, the modal responses can be combined and stored directly in the `Jobname.rst` file during spectrum solution according to the mode combination method command issued (**SRSS (p. 1845)**, **CQC (p. 433)**, etc.). This can save significant time compared to the method for *Elcalc* = NO, which requires generating the file of POST1 commands (`Jobname.mcom` file) to be read in POST1 to perform the mode combinations. For details and example usage, see Spectrum Analysis in the *Structural Analysis Guide*, Example 6.2: Calculating the Velocity Solution in Addition to the Displacement Solution with *Elcalc* = YES on **SPOPT** command, and Example: Multi-Point Response Spectrum (MPRS) Analysis.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When multiple response spectra solutions are calculated in SPRS or DDAM analyses with *Elcalc* = YES on the **SPOPT (p. 1837)** command:

- When missing mass is disabled (**MMASS (p. 1131)**):
  - If *SpecCum* = NO, each new response spectrum set overwrites the previous one at each solve in `Jobname.rst`.
  - If *SpecCum* = YES, all response spectra are cumulated in `Jobname.rst`.
- When missing mass is activated (**MMASS (p. 1131)**,ON) in a SPRS analysis:
  - If *SpecCum* = NO, the missing mass response (loadstep 2) and the response spectrum sets are cumulated in `Jobname.rst` until the **FINISH (p. 758)** command is issued, which activates the deletion of any response spectrum set between two missing mass response sets. Only the last response spectrum set is kept in `Jobname.rst`. See Single-Point Response Spectrum Analysis on a Piping Structure with Excitation Along X, Y, and Z Directions, missing mass effect, *Elcalc* = YES and *SpecCum* = NO on **SPOPT** command in the *Structural Analysis Guide*.

This command is also valid in PREP7.

Example Usage

Example 6.1: Calculating the Velocity Solution in Addition to the Displacement Solution

Example 6.2: Calculating the Velocity Solution in Addition to the Displacement Solution with *Elcalc* = YES on **SPOPT** command

6.4.1: Single-Point Response Spectrum Analysis on a Beam Structure

6.4.2: Single-Point Response Spectrum Analysis on a Piping Structure with Excitation Along X, Y, and Z Directions

6.4.3: Single-Point Response Spectrum Analysis on a Piping Structure with Excitation along X, Y, and Z Directions Separately by Reusing the Existing Mode File

6.4.4: Single-Point Response Spectrum Analysis on a Piping Structure with Excitation Along X, Y, and Z Directions, missing mass effect, *Elcalc* = YES and *SpecCum* = NO on **SPOPT** command

6.4.5: Single-Point Response Spectrum Analysis on a Piping Structure with Excitation Along X, Y, and Z Directions, missing mass effect, Elcalc = YES and *SpecCum* = YES on **SPOPT** command: method 1

6.4.6: Single-Point Response Spectrum Analysis on a Piping Structure with Excitation Along X, Y, and Z Directions, missing mass effect, *Elcalc* = YES on **SPOPT** command: method 2

Example: Random Vibration (PSD) Analysis

Example: DDAM Spectrum Analysis

Example: Multi-Point Response Spectrum (MPRS) Analysis

Where to Find Other Examples

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

SPREAD, *VALUE*

Turns on a dashed tolerance curve for the subsequent curve plots.

POST26 (p. 58): Display (p. 60)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VALUE

Amount of tolerance. For example, 0.1 is ± 10%.

Command Default

No tolerance curve.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

SPTOPT

Specifies "Spectrum analysis options" as the subsequent status topic.

SOLUTION (p. 38): Status (p. 49)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Show Status

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Show Status

Utility Menu>List>Status>Solution>Spectrum Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SPUNIT, *TBLNO*, *Type*, *GVALUE*, *KeyInterp*

Defines the type of multi-point response spectrum.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Input table number.

Type

Label identifying the type of spectrum:

DISP

Displacement spectrum (**SPVAL (p. 1842)** values interpreted as displacements with units of length).

VELO

Velocity spectrum (**SPVAL (p. 1842)** values interpreted as velocities with units of length/time).

ACEL

Acceleration spectrum (**SPVAL (p. 1842)** values interpreted as accelerations with units of length/time^2).

ACCG

Acceleration spectrum (**SPVAL (p. 1842)** values interpreted as accelerations with units of g/time^2).

FORC

Force spectrum.

PRES

Pressure spectrum.

GVALUE

Value of acceleration due to gravity in any arbitrary units for Type=ACCG table. Default is 386.4 in/sec^2.

KeyInterp

Key to activate or deactivate the linear interpolation between input response spectrum points and input response spectrum curves:

0 (OFF or NO)

Deactivate linear and use logarithmic interpolation. This value is the default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 (ON or YES)

Activate linear interpolation.

Command Default

Acceleration (ACEL) spectrum (length/time^2).

Notes

Defines the type of multi-point response spectrum defined by the **SPFREQ (p. 1828)** and **SPVAL (p. 1842)** commands.

Force (**FORC**) and pressure (**PRES**) type spectra can be used only as a nodal excitation.

GVALUE is valid only when *Type* = ACCG is specified. A zero or negative value cannot be used. A parameter substitution can also be performed.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Settings

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Settings

SPVAL, *TBLNO*, *CURVNO*, *SV1*, *SV2*, *SV3*, *SV4*, *SV5*, *SV6*, *SV7*

Defines multi-point response spectrum values.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TBLNO

Input table number. It corresponds to *TBLNO* on the **SPFREQ (p. 1828)** command.

CURVNO

Input curve number. It corresponds to *CURVNO* on the **SPDAMP (p. 1827)** command (optional).

SV1, SV2, SV3, , , . . . , SV7

Spectral values corresponding to the frequency points (**SPFREQ (p. 1828)**) and damping ratio (**SPDAMP (p. 1827)**). Values are interpreted as defined with the **SPUNIT (p. 1841)** command.

Notes

Defines multi-point response spectrum values to be associated with the previously defined frequency points (**SPFREQ (p. 1828)**). It can also be associated with the previously defined damping value (**SPDAMP (p. 1827)**). If *CURVNO* is not specified, the input spectrum is not associated with a damping value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Repeat **SPVAL (p. 1842)** command for additional values, up to the number of frequency points (**SPFREQ (p. 1828)**). Values are added after the last nonzero value.

The interpolation method between response spectrum points and curves is specified using *KeyInterp* on the **SPUNIT (p. 1841)** command. It is logarithmic by default.

Use the **SPTOPT (p. 1840)** and **STAT (p. 1861)** commands to list current spectrum curve values.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Spect vs Freq

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Spect vs Freq

---

SQRT, *IR*, *IA*, - - , - - , *Name*, - - , - - , *FACTA*

Forms the square root of a variable.

POST26 (p. 58): Operations (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Arbitrary reference number assigned to the resulting variable (2 to *NV* (**NUMVAR (p. 1302)**)). If this number is the same as for a previously defined variable, the previously defined variable will be overwritten with this result.

IA

Reference number of the variable to be operated on.

- -, - -

Unused fields.

Name

Thirty-two character name identifying the variable on printouts and displays. Embedded blanks are compressed for output.

- -, - -

Unused fields.

FACTA

Scaling factor (positive or negative) applied to variable *IA* (defaults to 1.0).

Notes

Forms the square root of a variable according to the operation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$IR = \sqrt{FACTA \times IA}$$

Menu Paths

Main Menu>TimeHist Postpro>Math Operations>Square Root

---

*SREAD,*StrArray*,*Fname*,*Ext*,*--*,*nChar*,*nSkip*,*nRead*

Reads a file into a string array parameter.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

StrArray

Name of the "string array" parameter which will hold the read file. String array parameters are similar to character arrays, but each array element can be as long as 248 characters. If the string parameter does not exist, it will be created. The array will be created as: *DIM,StrArray,STRING,nChar,nRead

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

--

Unused field.

nChar

Number of characters per line to read (default is length of the longest line in the file).

nSkip

Number of lines to skip at the start of the file (default is 0).

nRead

Number of lines to read from the file (default is the entire file).

Notes

The ***SREAD (p. 1844)** command reads from a file into a string array parameter. The file must be an ASCII text file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

SRSS,*SIGNIF*,*Label*,*AbsSumKey*,*ForceType*

Specifies the square root of sum of squares mode combination method.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SIGNIF

Combine only those modes whose significance level exceeds the *SIGNIF* threshold. For single-point, multi-point, or DDAM response spectrum (*Spectype* = SPRS, MPRS, or DDAM on **SPOPT (p. 1837)**), the significance level of a mode is defined as the mode coefficient divided by the maximum mode coefficient of all modes. Any mode whose significance level is less than *SIGNIF* is considered insignificant and is not contributed to the mode combinations. The higher the *SIGNIF* threshold, the fewer the number of modes combined. *SIGNIF* defaults to 0.001. If *SIGNIF* is specified as 0.0, it is taken as 0.0. (This mode combination method is not valid for **SPOPT (p. 1837)**,PSD.)

Label

Label identifying the combined mode solution output.

DISP

Displacement solution (default). Displacements, stresses, forces, etc., are available.

VELO

Velocity solution. Velocities, "stress velocities," "force velocities," etc., are available.

ACEL

Acceleration solution. Accelerations, "stress accelerations," "force accelerations," etc., are available.

AbsSumKey

Absolute Sum combination key (for **SPOPT (p. 1837)**,MPRS only):

NO

Do not use the Absolute Sum method (default).

YES

Combine the modes per excitation direction using the Absolute Sum method, then combine the resulting quantities using the square root of sum of squares method.

When using Absolute Sum combination, the excitation direction must be specified using the **SED (p. 1704)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ForceType

Label identifying the forces to be combined:

STATIC

Combine the modal static forces (default).

TOTAL

Combine the modal static plus inertial forces.

Notes

This command is also valid for PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>Mode Combine>SRSS Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>MultiPt>Mode Combine>SRSS Method

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Mode Combine>SRSS Method

Main Menu>Solution>Load Step Opts>Spectrum>Mode Combine>SRSS Method

Main Menu>Solution>Load Step Opts>Spectrum>MultiPt>Mode Combine>SRSS Method

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Mode Combine>SRSS Method

---

SSBT, B^T_{11}, B^T_{22}, B^T_{12}, T

Specifies preintegrated bending thermal effects for shell sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

B^T_{11}, B^T_{22}, B^T_{12}

Bending thermal effects component [B^T].

T

Temperature.

Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$
$$[S] = [E]\{\gamma\}$$

The **SSBT (p. 1846)** command, one of several preintegrated shell section commands, specifies the bending thermal effects quantity (submatrix [$\boldsymbol{B}^T$] data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

The [$\boldsymbol{B}^T$] quantity represents bending stress resultants caused by a unit raise in temperature on a fully constrained model. For a layered composite shell, it is usually necessary to specify both the [$\boldsymbol{B}^T$] and [$\boldsymbol{M}^T$] quantities (by issuing the **SSBT (p. 1846)** and **SSMT (p. 1849)** commands, respectively).

Unspecified values default to zero.

Related commands are **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, and **SSPM (p. 1855)**.

If you are using the SHELL181 or SHELL281 element's Membrane option (KEYOPT(1) = 1), it is not necessary to issue this command.

For complete information, see Creating a Preintegrated General Shell Section.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

/SSCALE, *WN*, *SMULT*

Sets the contour multiplier for topographic displays.

GRAPHICS (p. 16): Scaling (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

SMULT

Contour multiplier that factors in results based on the product of the multiplier and the result being plotted. Defaults to 0.0 (no topographic effects).

Command Default

No topographic contour effects.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Use this command to scale values to the geometry when the contours are shown elevated. For section displays (**/TYPE (p. 2034)**), the elevation is performed perpendicular to the section face.

Nonzero contour multipliers factoring in large results (stresses or displacements) can produce very large distortion, causing images to disappear. To bring a distorted image back into view, reduce the contour multiplier value.

Portions of this command are not supported by PowerGraphics (**/GRAPHICS (p. 866)**,POWER).

Menu Paths

Utility Menu>PlotCtrls>Style>Contours>Contour Style

SSLN, *FACT*, *SIZE*

Selects and displays small lines in the model.

PREP7 (p. 22): Lines (p. 26)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FACT

Factor used to determine small lines. *FACT* times the average line length in the model is used as the line length limit below which lines will be selected.

SIZE

Line length limit for line selection. Lines that have a length less than or equal to *SIZE* will be selected. Used only if *FACT* is blank.

Notes

SSLN (p. 1848) invokes a predefined Mechanical APDL macro for selecting small lines in a model. Lines that are smaller than or equal to the specified limit (*FACT* or *SIZE*) are selected and line numbers are displayed. This command macro is useful for detecting very small lines in a model that may cause problems (that is, poorly shaped elements or a meshing failure) during meshing. All lines that are not "small" will be unselected and can be reselected with the **LSEL (p. 1074)** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Sel Small Lines

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SSMT, M^T_{11}, M^T_{22}, M^T_{12}, T

Specifies preintegrated membrane thermal effects for shell sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

M^T_{11}, M^T_{22}, M^T_{12}

Membrane thermal effects component [***M^T***].

T

Temperature.

Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$

$$[S] = [E]\{\gamma\}$$

The **SSMT (p. 1849)** command, one of several preintegrated shell section commands, specifies the membrane thermal effects quantity (submatrix [***M^T***] data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

The [***M^T***] quantity represents membrane stress resultants caused by a unit raise in temperature on a fully constrained model. For a layered composite shell, it is usually necessary to specify both the [***M^T***] and [***B^T***] quantities (by issuing the **SSMT (p. 1849)** and **SSBT (p. 1846)** commands, respectively).

Unspecified values default to zero.

Related commands are **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSBT (p. 1846)**, and **SSPM (p. 1855)**.

For complete information, see Creating a Preintegrated General Shell Section.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SSOPT,*Option*,*Par1*,*Par2*,*Par3*,*Par4*,*Par5*

Defines a solution option for soil analysis.

SOLUTION (p. 38): Nonlinear Options (p. 40)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Solution option to define:

GEOSTATIC

Geostatic equilibrium step for soil analysis.

CONSOLIDATION

Consolidation step for soil analysis.

STOP

Stop condition for soil consolidation analysis.

SFSW

Specific weight load.

Par1, ... , Par5

Parameters for the specified *Option*.

Valid *Par* Values for Each *Option*

Option = GEOSTATIC

No parameter values required.

Option = CONSOLIDATION

No parameter values required.

Option = STOP

Par1:

SSTATE -- The steady-state solution threshold of incremental pore pressure in a step.

OFF -- Deactivate steady-state solution check.

Par2:

Valid only when *Par1* = SSTATE.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A positive value to define the maximum pore pressure increment in a step, or a negative value to define the percentage of incremental pore pressure in a step to maximum pore pressure in the solution.

Option **= SFSW**

Par1, Par2, Par3:

The specific weight load direction. (*Default*: The -Y axis in the global coordinate system.)

Par4:

OFF -- Ignore the specific bulk weight (default).

ON -- Account for the specific bulk weight load.

Par5:

OFF -- Ignores the fluid specific weight (default).

ON -- Account for the fluid specific weight.

Notes

The **SSOPT (p. 1850)** command defines solution options for soil analysis (**ANTYPE (p. 162)**,SOIL) only.

Menu Paths

This command cannot be accessed from a menu.

SSPA,A_{11} ,A_{21} ,A_{31} ,A_{22} ,A_{32} ,A_{33} ,T

Specifies a preintegrated membrane stiffness for shell sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

A_{11} ,A_{21} ,A_{31} ,A_{22} ,A_{32} ,A_{33}

Membrane stiffness component (symmetric lower part of submatrix [***A***]).

T

Temperature.

Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$
$$[S] = [E]\{\gamma\}$$

The **SSPA (p. 1851)** command, one of several preintegrated shell section commands, specifies the membrane stiffness quantity (submatrix [***A***]) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero.

Related commands are **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, **SSBT (p. 1846)**, and **SSPM (p. 1855)**.

For complete information, see Creating a Preintegrated General Shell Section.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

SSPB, B_{11} , B_{21} , B_{31} , B_{22} , B_{32} , B_{33} , T, B_{12} , B_{13} , B_{23}

Specifies a preintegrated coupling stiffness for shell sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

B_{11} , B_{21} , B_{31} , B_{22} , B_{32} , B_{33}

Coupling stiffness component (symmetric lower part of submatrix [***B***]).

T

Temperature.

B_{12} , B_{13} , B_{23}

Upper part of submatrix [***B***]

Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$
$$[S] = [E]\{\gamma\}$$

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the coefficients B_{12} , B_{13} , B_{23} are undefined, the program uses a symmetric form of submatrix [***B***]. If any one of the coefficients B_{12} , B_{13} , B_{23} is nonzero, the program considers submatrix [***B***] to be unsymmetric.

The **SSPB (p. 1852)** command, one of several preintegrated shell section commands, specifies the coupling stiffness quantity (submatrix [***B***] data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero.

Related commands are **SSPA (p. 1851)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, **SSBT (p. 1846)**, and **SSPM (p. 1855)**.

If you are using the SHELL181 or SHELL281 element's Membrane option (KEYOPT(1) = 1), it is not necessary to issue this command.

For complete information, see Creating a Preintegrated General Shell Section.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

SSPD, D_{11} , D_{21} , D_{31} , D_{22} , D_{32} , D_{33} , T

Specifies a preintegrated bending stiffness for shell sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

D_{11} , D_{21} , D_{31} , D_{22} , D_{32} , D_{33}

Bending stiffness component (symmetric lower part of submatrix [***D***]).

T

Temperature.

Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$

$$[S] = [E]\{\gamma\}$$

The **SSPD (p. 1853)** command, one of several preintegrated shell section commands , specifies the bending stiffness quantity (submatrix [***D***] data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unspecified commands default to zero.

Related commands are **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, **SSBT (p. 1846)**, and **SSPM (p. 1855)**.

If you are using the SHELL181 or SHELL281 element's Membrane option (KEYOPT(1) = 1), it is not necessary to issue this command.

For complete information, see Creating a Preintegrated General Shell Section.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

SSPE, E_{11} , E_{21} , E_{22} , T

Specifies a preintegrated transverse shear stiffness for shell sections.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

E_{11} , E_{21} , E_{22}

Transverse shear stiffness component (symmetric lower part of submatrix [***E***]).

T

Temperature.

Notes

The behavior of shell elements is governed by the generalized-stress/generalized-strain relationship of the form:

$$\begin{Bmatrix} N \\ M \end{Bmatrix} = \begin{bmatrix} A & B \\ B^T & D \end{bmatrix} \begin{Bmatrix} \varepsilon \\ \kappa \end{Bmatrix} - (T - T^I) \begin{Bmatrix} M^T \\ B^T \end{Bmatrix}$$

$$[S] = [E]\{\gamma\}$$

The **SSPE (p. 1854)** command, one of several preintegrated shell section commands, specifies the transverse shear stiffness quantity (submatrix [***E***] data) for a preintegrated shell section. The section data defined is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Unspecified values default to zero.

Related commands are **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSMT (p. 1849)**, **SSBT (p. 1846)**, and **SSPM (p. 1855)**.

If you are using the SHELL181 or SHELL281 element's Membrane option (KEYOPT(1) = 1), it is not necessary to issue this command.

For complete information, see Creating a Preintegrated General Shell Section .

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

SSPM,*DENS*,*T*

Specifies mass density for a preintegrated shell section.

PREP7 (p. 22): Cross Sections (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DENS

Mass density.

T

Temperature.

Notes

The **SSPM (p. 1855)** command, one of several preintegrated shell section commands, specifies the mass density (assuming a unit thickness) for a preintegrated shell section. The value specified is associated with the section most recently defined (via the **SECTYPE (p. 1697)** command).

Related commands are **SSPA (p. 1851)**, **SSPB (p. 1852)**, **SSPD (p. 1853)**, **SSPE (p. 1854)**, **SSMT (p. 1849)**, and **SSBT (p. 1846)**.

For complete information, see Creating a Preintegrated General Shell Section.

Menu Paths

Main Menu>Preprocessor>Sections>Shell>Pre-integrated

SSTATE,*Action*,*CM_Name*,*Val1*,*Val2*,*Val3*,*Val4*,*Val5*,*Val6*,*Val7*,*Val8*,*Val9*

Defines a steady-state rolling analysis.

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Action to perform for defining or manipulating steady-state rolling analysis data:

DEFINE

Define steady-state rolling analysis data

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LIST

List current steady-state rolling analysis data

DELETE

Delete steady-state rolling analysis data

CM_Name

Element component name

VAL1, ..., VAL9

Input values (based on the *Action* type)

Notes

The **SSTATE (p. 1855)** command specifies steady-state rolling analysis parameters for the given element component. The program runs the steady-state rolling analysis if the corresponding element key option is enabled for that element component.

The command supports the following elements:

Supported Element	Description
SOLID185	3D 8-Node Structural Solid
SOLID186	3D 20-Node Structural Solid
SOLID187	3D 10-Node Tetrahedral Structural Solid
SOLSH190	3D 8-Node Structural Solid Shell Degenerated shape (prism) option not supported.
SOLID285	3D 4-Node Tetrahedral Structural Solid with Nodal Pressures

For information about steady-state rolling for rebar and solid elements, see Steady-State Rolling in the *Mechanical APDL Theory Reference*.

Command Specification for *Action* = DEFINE

The following data types can be defined:

- SPIN -- Spinning motion
- TRANSLATE -- Rigid body motion (velocity) that the spinning component is undergoing

Define the steady-state spinning motion:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SSTATE (p. 1855), DEFINE, *CM_Name*, SPIN, *OMEGA*, *Method*, *Val4*, *Val5*, *Val6*, *Val7*, *Val8*, *Val9*

OMEGA

Spin velocity

Method

Method to use for defining the spin axis:

POINTS --

Define the spin axis using two points:

Val4, *Val5*, *Val6* -- Coordinates of the first point

Val7, *Val8*, *Val9* -- Coordinates of the second point

This definition method is currently the only option.

Example 18: Defining Steady-State Spinning Motion

This command defines a steady-state spinning motion of 120 rad/s around the spin axis:

```
SSTATE,DEFINE,CM_Name,SPIN,120,POINTS,0,0,0,0,1,0
```

In this case, two points with coordinates (0,0,0) and (0,1,0) define the spin axis in the global Y direction.

Define the rigid body motion (velocity):

SSTATE (p. 1855), DEFINE, *CM_Name*, TRANSLATE, *Val2*, *Val3*, *Val4*

Val2, *Val3*, *Val4* -- Rigid body velocity components

Command Specification for *Action* = LIST

SSTATE (p. 1855), LIST, *CM_Name*

Lists all steady-state rolling analysis data defined on the specified element component. All data is listed if no component (*CM_Name*) is specified.

Command Specification for *Action* = DELETE

SSTATE (p. 1855), DELETE, *CM_Name*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deletes all steady-state rolling analysis data defined on the specified element component. All data is deleted if no component (*CM_Name*) is specified.

Menu Paths

This command cannot be accessed from a menu.

SSUM

Calculates and prints the sum of element table items.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Calculates and prints the tabular sum of each existing labeled result item (**ETABLE (p. 687)**) for the selected elements. If absolute values are requested (**SABS (p. 1643)**,1), absolute values are used.

Menu Paths

Main Menu>General Postproc>Element Table>Sum of Each Item

STABILIZE,*Key*,*Method*,*VALUE*,*SubStpOpt*,*FORCELIMIT*,*RECALCDAMP*

Activates stabilization for all elements that support nonlinear stabilization.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Key for controlling nonlinear stabilization:

OFF

Deactivate stabilization (default).

CONSTANT

Activate stabilization. The energy-dissipation ratio or damping factor remains constant during the load step.

REDUCE

Activate stabilization. The energy-dissipation ratio or damping factor is reduced linearly to zero at the end of the load step from the specified or calculated value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method

The stabilization-control method:

ENERGY

Use the energy-dissipation ratio as the control. This value is the default when *Key* ≠ OFF.

DAMPING

Use the damping factor as the control.

VALUE

The energy-dissipation ratio (*Method* = ENERGY) or damping factor (*Method* = DAMPING). This value must be greater than 0 when *Method* = ENERGY or *Method* = DAMPING. When *Method* = ENERGY, this value is usually a number between 0 and 1.

SubStpOpt

Option for the first substep of the load step:

NO

Stabilization is not activated for the first substep even when it does not converge after the minimal allowed time increment is reached. This value is the default when *Key* ≠ OFF.

MINTIME

Stabilization is activated for the first substep if it still does not converge after the minimal allowed time increment is reached.

ANYTIME

Stabilization is activated for the first substep. Use this option if stabilization was active for the previous load step via *Key* = CONSTANT.

FORCELIMIT

The stabilization force limit coefficient, such that 0 < *FORCELIMIT* < 1. The default value is 0.2. To omit a stabilization force check, set this value to 0.

RECALCDAMP

Key for controlling damping recalculation:

0 – No recalculation of the damping factor (default).

1 – Recalculate the damping factor for the energy-based stabilization-control method.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Once issued, the stabilization effects of the command remain until you issue either a **STABILIZE (p. 1858)** command (with no arguments) or a **STABILIZE (p. 1858)**,OFF command. If you issue the command with no arguments, the effect is to deactivate stabilization.

Notes

Once issued, a **STABILIZE (p. 1858)** command remains in effect until you reissue the command.

For the energy dissipation ratio, specify *VALUE* = 1.0e-4 if you have no prior experience with the current model; if convergence problems are still an issue, increase the value gradually. The damping factor is mesh-, material-, and time-step-dependent; an initial reference value from the previous run (such as a run with the energy-dissipation ratio as input) should suggest itself.

Exercise caution when specifying *SubStpOpt* = MINTIME or ANYTIME for the first load step; Ansys, Inc. recommends this option only for experienced users. If stabilization was active for the previous load step via *Key* = CONSTANT and convergence is an issue for the first substep, specify *SubStpOpt* = ANYTIME.

When the L2-norm of the stabilization force (CSRSS value) exceeds the L2-norm of the internal force multiplied by the stabilization force coefficient, the program issues a message displaying both the stabilization force norm and the internal force norm. The *FORCELIMIT* argument enables you to change the default stabilization force coefficient (normally 20 percent).

When using the energy-based stabilization-control method and *RECALCDAMP* = 1, the damping factor is recalculated in the following cases:

- In an analysis restart.
- In a nonlinear adaptivity analysis in the substep following the remeshing substep.

This command stabilizes the degrees of freedom for current-technology elements only. Other elements can be included in the FE model, but their degrees of freedom are not stabilized.

For more information about nonlinear stabilization, see Unstable Structures in the *Structural Analysis Guide*. For additional tips that can help you to achieve a stable final model, see Simplify Your Model in the *Structural Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Advanced NL

Main Menu>Solution>Analysis Type>Sol'n Controls>Advanced NL

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STAT

Displays the status of database settings.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

STAT (p. 1861) is a command generated by the GUI and will appear in the log file (`Jobname.log`) if status is requested for some items under **Utility Menu> List> Status**. Generally, **STAT (p. 1861)** will be preceded by one of the commands listed below, which specifies the particular topic that status was requested for.

If entered directly into the program, the **STAT (p. 1861)** command should be immediately preceded by the desired topic command listed below. In processors other than those listed below (for example, AUX12), no topic command should proceed **STAT (p. 1861)**.

This command is valid in any processor.

PREP7 topic commands (and their corresponding topics) are:

Topic Command	Topic
ETYPE (p. 703)	Element types
RCON (p. 1574)	Real constants
MATER (p. 1112)	Material properties
TBLE (p. 1988)	Data table properties
PRIM (p. 1495)	Solid model primitives
KEYPTS (p. 980)	Keypoints
LINE (p. 1051)	Lines
AREAS (p. 178)	Areas
VOLUMES (p. 2141)	Volumes
GEOMETRY (p. 797)	Solid model information
MESHING (p. 1124)	Meshing
BOOL (p. 268)	Booleans
NODES (p. 1248)	Nodes
ELEM (p. 618)	Elements
SELM (p. 1709)	Superelements
COUPLE (p. 420)	Node coupling
CEQN (p. 310)	Constraint equations

SOLUTION topic commands (and their corresponding topics) are:

Topic Command	Topic
ATYPE (p. 206)	Analysis types

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Topic Command	Topic
MASTER (p. 1111)	Master DOF
DEACT (p. 510)	Element birth and death (deactivation)
LSOPER (p. 1080)	Load step operations
FECONS (p. 745)	Constraints on nodes
FEFOR (p. 746)	Forces on nodes
FESURF (p. 746)	Surface loads on elements
FEBODY (p. 745)	Body loads on elements
SMCONS (p. 1807)	Constraints on the solid model
SMFOR (p. 1807)	Forces on the solid model
SMSURF (p. 1814)	Surface loads on the solid model
SMBODY (p. 1806)	Body loads on the solid model
INRTIA (p. 955)	Inertial loads
GENOPT (p. 796)	General options
DYNOPT (p. 590)	Dynamic analysis options
NLOPT (p. 1247)	Nonlinear analysis options
OUTOPT (p. 1332)	Output options
BIOOPT (p. 264)	Biot-Savart options
SPTOPT (p. 1840)	Spectrum analysis options
SOLUOPT (p. 1821)	Solution options

POST1 topic commands (and their corresponding topics) are:

Topic Command	Topic
DEFINE (p. 511)	Data definition settings
SORT (p. 1824)	Sort settings
PRINT (p. 1496)	Print settings
DISPLAY (p. 535)	Display settings
CALC (p. 283)	Calculation settings
PATH (p. 1354)	Path data settings
LCCALC (p. 1020)	Load case settings
DATADEF (p. 498)	Directly defined data status
POINT (p. 1459)	Point flow tracing settings
SPEC (p. 1828)	Miscellaneous specifications

POST26 topic commands (and their corresponding topics) are:

Topic Command	Topic
DEFINE (p. 511)	Data definition settings
OPERATE (p. 1325)	Operation data
PRINT (p. 1496)	Print settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Topic Command	Topic
PLOTTING (p. 1438)	Plotting settings

Menu Paths

Main Menu>Solution>Solve>Current LS

---

*STATUS, *Par*, *IMIN*, *IMAX*, *JMIN*, *JMAX*, *KMIN*, *KMAX*, *LMIN*, *LMAX*, *MMIN*, *MMAX*, *KPRI*

Lists the current parameters and abbreviations.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

Specifies the parameter or sets of parameters listed. For array parameters, use *IMIN*, *IMAX*, etc. to specify ranges. Use ***DIM (p. 530)** to define array parameters. Use ***VEDIT (p. 2089)** to review array parameters interactively. Use ***VWRITE (p. 2174)** to print array values in a formatted output. If *Par* is blank, list all scalar parameter values, array parameter dimensions, and abbreviations. If *ARGX*, list the active set of local macro parameters (ARG1 to ARG9 and AR10 to AR99) (***USE (p. 2059)**).

The following are possible values for *Par*

ALL or blank --

Lists all parameters (except local macro parameters and those with names beginning or ending with an underbar) and toolbar abbreviations.

_PRM --

Lists only parameters with names beginning with an underbar (_). These are Mechanical APDL internal parameters.

PRM_ --

Lists only parameters with names ending with an underbar (_). A good APDL programming convention is to ensure that all parameters created by your system programmer are named with a trailing underbar.

ABBR --

Lists all toolbar abbreviations.

PARM --

Lists all parameters (except local macro parameters and those with names beginning or ending with an underbar).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MATH --

Lists all APDL Math parameters, including vectors, matrices, and linear solvers.

***PARNAME* --**

Lists only the parameter specified. *PARNAME* cannot be a local macro parameter name.

***ARGX* --**

Lists all local macro parameter values (ARG1- AR99) that are non-zero or non-blank.

IMIN*, *IMAX*, *JMIN*, *JMAX*, *KMIN*, *KMAX*, *LMIN*, *LMAX*, *MMIN*, *MMAX

Range of array elements to display (in terms of the dimensions (row, column, plane, book, and shelf). Minimum values default to 1. Maximum values default to the maximum dimension values. Zero may be input for *IMIN*, *JMIN*, and *KMIN* to display the index numbers. See ***TAXIS (p. 1898)** command to list index numbers of 4- and 5-D tables.

KPRI

Use this field to list your primary variable labels (X, Y, Z, TIME, etc.).

1

List the labels (default). *YES*, *Y*, or *ON* are also valid entries.

0

Do not list the labels. *NO*, *N*, or *OFF* are also valid entries.

Notes

You cannot obtain the value for a single local parameter (for example, ***STATUS (p. 1863)**,ARG2). You can only request all local parameters simultaneously using ***STATUS (p. 1863)**,ARGX.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Other>Named Parameter

Utility Menu>List>Other>Parameters

Utility Menu>List>Status>Parameters>All Parameters

Utility Menu>List>Status>Parameters>Named Parameters

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/STATUS, *Lab*

Lists the status of items for the run.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Items to list status for:

ALL

List all below (default).

TITLE

List only titles, Jobname, and revision number.

UNITS

List only units.

MEM

List only memory data statistics.

DB

List only database statistics

CONFIG

List only configuration parameters.

GLOBAL

Provides a global status summary.

SOLU

Provides a solution status summary.

PROD

Provides a product summary.

Notes

Displays various items active for the run (such as the Mechanical APDL revision number, Jobname, titles, units, configuration parameters, database statistics, etc.).

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>List>Status>Global Status

STEF, *VALUE*

Specifies Stefan-Boltzmann radiation constant.

AUX12 (p. 62): General Radiation (p. 62)
SOLUTION (p. 38): Radiosity (p. 50)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VALUE

Stefan-Boltzmann constant (defaults to 0.119E-10 Btu/hr/in^2/ °R^4).

Command Default

0.119E-10 Btu/hr/in^2/ °R^4.

Notes

You can use this command in the general preprocessor (PREP7) and in the Solution processor to specify the Stefan-Boltzmann constant in analyses using the radiation matrix method or the radiosity solver to model radiation.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Areas
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Elements
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Lines
Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Ambient Rad>On Nodes
Main Menu>Preprocessor>Radiation Opts>Solution Opt
Main Menu>Radiation Opt>Matrix Method>Other Settings
Main Menu>Radiation Opt>Radiosity Meth>Solution Opt
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Areas
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Elements
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Lines
Main Menu>Solution>Define Loads>Apply>Thermal>Ambient Rad>On Nodes
Main Menu>Solution>Radiation Opts>Solution Opt

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/STITLE, *NLINE*, *Title*

Defines subtitles.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NLINE

Subtitle line number (1 to 4). Defaults to 1.

Title

Input up to 70 alphanumeric characters. Parameter substitution may be forced within the title by enclosing the parameter name or parametric expression within percent (%) signs. If *Title* is blank, this subtitle is deleted.

Notes

Up to four subtitles are displayed in the output along with the main title (**/TITLE (p. 2007)**).

Subtitles do not appear in GUI windows or in plot displays.

The first subtitle is also written to various Mechanical APDL files along with the main title.

Previous subtitles can be overwritten or deleted.

Issue **/STATUS (p. 1865)** to display titles.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

STORE, *Lab*, *NPTS*, *--*, *FREQ*, *Toler*, *Cluster*

Stores data in the database for the defined variables.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Valid labels:

MERGE

Merge data from results file for the time points in memory with the existing data using current specifications (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NEW

Store a new set of data, replacing any previously stored data with current result file specifications and deleting any previously-calculated variables (see Table 2.88: Operations (p. 59)).

Variables defined using the **ANSOL (p. 149)** command are also deleted if they represent element-based results. Variables created using nodal-averaged results are not deleted.

APPEN

Append data from results file to the existing data.

ALLOC

Allocate (and zero) space for *NPTS* data points.

PSD

Create a new set of frequency points for PSD calculations (replacing any previously stored data and erasing any previously calculated data).

NPTS

The number of time points (or frequency points) for storage (used only with *Lab* = ALLOC or PSD). The value may be input when using POST26 with data supplied from other than a results file. This value is automatically determined from the results file data with the NEW, APPEN, and MERGE options. For the PSD option, *NPTS* determines the resolution of the frequency vector (valid numbers are between 1 and 10, defaults to 5).

--

Unused field.

FREQ

A frequency value, or an array containing frequency values (Hz). Use ***DIM (p. 530)** to define the array and enclose the array name in percent signs (for example, **STORE (p. 1867)**,,,,%*arrayname*%). A default value of 1% of damping is considered for clustering around the user-input frequency values. Supported for *Lab* = PSD only.

Toler

Tolerance to determine if a user-input frequency value (*FREQ*) is a duplicate and can be ignored. Two frequency values are considered duplicates if their difference is smaller than the frequency range multiplied by the tolerance. The default value is 10^{-5}. Supported for *Lab* = PSD and *Cluster* = YES only.

Cluster

Key to control whether or not to consider the clustering frequencies around each of the user-input array values. Available only when a user-defined frequency array is used (*FREQ*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

YES

Merge the clustering frequencies around both the natural frequencies and the frequency values entered in the user-defined array (*FREQ*) (default).

NO

Do not include clustering frequencies, and use only natural frequencies and the frequencies in the user-defined array (*FREQ*).

Command Default

Merge newly-defined variables with previously stored variables for the time points stored in memory using the current specifications. If **STORE (p. 1867)** is preceded by **TIMERANGE (p. 2001)** or **NSTORE (p. 1287)**, the default is **STORE (p. 1867)**,NEW.

Notes

This command stores data from the results file in the database for the defined variables (**ANSOL (p. 149)**, **NSOL (p. 1281)**, **ESOL (p. 672)**, **SOLU (p. 1819)**, **JSOL (p. 966)**) per specification (**AVPRIN (p. 210)**, **FORCE (p. 772)**, **LAYERP26 (p. 1016)**, **SHELL (p. 1784)**). See Storing the Variable in the *Basic Analysis Guide* for more information.

The **STORE (p. 1867)**,PSD command creates a new frequency vector (variable 1) for response PSD calculations (**RPSD (p. 1619)**). This command should first be issued before defining variables (**NSOL (p. 1281)**, **ESOL (p. 672)**, **RFORCE (p. 1607)**) for which response PSD's are to be calculated.

If the frequencies in the user-defined array are relevant, turning off clustering (*Cluster* = NO) reduces calculation costs without significant loss of accuracy. You can check the frequencies by initially issuing a default **RPSD (p. 1619)** (with clustering) to obtain a reference plot of the response.

Menu Paths

Main Menu>TimeHist Postpro>Store Data

SUBOPT,*Option*,*Value1*

Specifies Subspace (SUBSP) eigensolver options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

One of the following options:

STRMCK

Controls whether a Sturm sequence check is performed.

Valid input for *Value1* when *Option* = STRMCK:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Do not perform Sturm sequence check (default).

ON

Perform Sturm sequence check.

MEMORY

Controls the memory allocation strategy for the Subspace eigensolver.

Valid input for *Value1* when *Option* = MEMORY:

DEFAULT

Default memory configuration (default). Everything is determined dynamically with respect to current machine resources.

INCORE

Fully in-core memory configuration.

MIX1

First level of mixed in-core / out-of-core configuration.

MIX2

Second level of mixed in-core / out-of-core configuration.

OUTOFCORE

Fully out-of-core memory configuration.

Value1

Assigned value for the specified *Option* (as described above).

Command Default

The Sturm sequence check is off for the Supspace eigensolver.

The default memory allocation strategy is used. Mechanical APDL evaluates the resources of the machine to choose the in-core or out-of-core mode.

Notes

SUBOPT (p. 1869) specifies options to be used with the Subspace eigensolver (**MODOPT (p. 1140)**,SUBSP) during a modal analysis.

Memory Allocation Option (*Option* = MEMORY)

The Subspace eigensolver algorithm allocates two main pools of memory:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Memory for the internal subspace eigensolver iterations.
2. Memory for the specific subspace eigensolver working arrays.

The following table shows how memory is allocated for each option.

Value1 **(*Option* = MEMORY)**	**Subspace Eigensolver**	**Working Arrays**
INCORE	In-Core	In-Core
MIX1	In-Core	Out-of-Core
MIX2	Out-of-Core	In-Core
OUTOFCORE	Out-of-Core	Out-of-Core

The MIX1 configuration typically uses more memory than the MIX2 configuration, except when a large number of modes are requested for a small model.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

SUBSET,*Lstep*,*SBSTEP*,*FACT*,*KIMG*,*TIME*,*ANGLE*,*NSET*

Reads results for the selected portions of the model.

POST1 (p. 51): Set Up (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lstep

Load step number of the data set to be read (defaults to 1):

N

Read load step *N*.

FIRST

Read the first data set (*SBSTEP* and *TIME* are ignored).

LAST

Read the last data set (*SBSTEP* and *TIME* are ignored).

NEXT

Read the next data set (*SBSTEP* and *TIME* are ignored). If at the last data set, the first data set will be read as the next.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NEAR

Read the data set nearest to *TIME* (*SBSTEP* is ignored). If *TIME* is blank, read the first data set.

LIST

Scan the results file and list a summary of each load step. (*FACT*, *KIMG*, *TIME* and *ANGLE* are ignored.)

SBSTEP

Substep number (within *Lstep*). For the buckling (**ANTYPE (p. 162)**,BUCKLE) analysis or the modal (**ANTYPE (p. 162)**,MODAL) analysis, the substep corresponds to the mode number. Defaults to last substep of load step (except for **ANTYPE (p. 162)**,BUCKLE or MODAL). If *Lstep* = LIST, *SBSTEP* = 0 or 1 lists the basic step information, whereas *SBSTEP* = 2 also lists the load step title, and labels imaginary data sets if they exist.

FACT

Scale factor applied to data read from the file. If zero (or blank), a value of 1.0 is used. Harmonic velocities or accelerations may be calculated from the displacement results from a modal (**ANTYPE (p. 162)**,MODAL) or harmonic (**ANTYPE (p. 162)**,HARMIC) analyses. If *FACT* = VELO, the harmonic velocities (v) are calculated from the displacements (d) at a particular frequency (f) according to the relationship $v = 2 \pi f d$. Similarly, if *FACT* = ACEL, the harmonic accelerations (a) are calculated as $a = (2 \pi f)^2 d$.

KIMG

Used only with results from complex analyses:

0

Store real part of complex solution

1

Store imaginary part.

TIME

Time-point identifying the data set to be read. For harmonic analyses, time corresponds to the frequency. For the buckling analysis, time corresponds to the load factor. Used only in the following cases: If *Lstep* is NEAR, read the data set nearest to *TIME*. If both *Lstep* and *SBSTEP* are zero (or blank), read data set at time = *TIME*. If *TIME* is between two solution time points on the results file, a linear interpolation is done between the two data sets. Solution items not written to the results file (**OUTRES (p. 1336)**) for either data set will result in a null item after data set interpolation. If *TIME* is beyond the last time point on the file, use the last time point.

ANGLE

Circumferential location (0.0 to 360°). Defines the circumferential location for the harmonic calculations used when reading from the results file. The harmonic factor (based on the circumferential angle) is applied to the harmonic elements (PLANE25, PLANE75, PLANE78, PLANE83, and SHELL61)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the load case. See the *Mechanical APDL Theory Reference* for details. Note that factored values of applied constraints and loads will overwrite any values existing in the database.

NSET

Data set number of the data set to be read. If a positive value for *NSET* is entered, *Lstep*, *SBSTEP*, *KIMG*, and *TIME* are ignored. Available set numbers can be determined by **SET (p. 1724)**,LIST.

Notes

Reads a data set from the results file into the database for the selected portions of the model only. Data that has not been specified for retrieval from the results file by the **INRES (p. 953)** command will be listed as having a zero value. Each time that the **SUBSET (p. 1871)** command is issued, the data currently in the database will be overwritten with a new set of data. Various operations may also be performed during the read operation. The database must have the model geometry available (or used the **RESUME (p. 1601)** command before the **SUBSET (p. 1871)** command to restore the geometry from File.db).

Menu Paths

Main Menu>General Postproc>Read Results>By Load Step

Main Menu>General Postproc>Read Results>By Set Number

Main Menu>General Postproc>Read Results>By Time/Freq

SUCALC,*RSetName*,*lab1*,*Oper*,*lab2*,*fact1*,*fact2*,*const*

Create new result data by operating on two existing result data sets on a given surface.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RSetName

Eight character name for new result data.

lab1

First result data upon which to operate.

Oper

Mathematical operation to perform.

ADD

(*lab1* + *lab2* + *const*)

SUB

(*lab1* - *lab2* + *const*)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MULT

(*lab1* * *lab2* + *const*)

DIV

(*lab1* / *lab2* + *const*)

EXP

(*lab1* ^ *fact1* + *lab2* ^ *fact2* + *const*)

COS

(cos (*lab1*) + *const*)

SIN

(sin (*lab1*) + *const*)

ACOS

(acos (*lab1*) + *const*)

ASIN

(asin (*lab1*) + *const*)

ATAN

(atan (*lab1*) + *const*)

ATA2

(atan2 (*lab1* / *lab2*) + *const*)

LOG

(log (*lab1*) + *const*)

ABS

(abs (*lab1*) + *const*)

ZERO

(0 + *const*)

lab2

Second result data upon which to operate.

fact1

First scaling factor (for EXP option only).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fact2

Second scaling factor (for EXP option only).

const

Constant added to the values in the resulting data.

Menu Paths

Main Menu>General Postproc>Surface Operations>Math Operations>Absolute
Main Menu>General Postproc>Surface Operations>Math Operations>Add
Main Menu>General Postproc>Surface Operations>Math Operations>ArcCosine
Main Menu>General Postproc>Surface Operations>Math Operations>ArcSine
Main Menu>General Postproc>Surface Operations>Math Operations>ArcTangent
Main Menu>General Postproc>Surface Operations>Math Operations>ArcTangent2
Main Menu>General Postproc>Surface Operations>Math Operations>Cosine
Main Menu>General Postproc>Surface Operations>Math Operations>Divide
Main Menu>General Postproc>Surface Operations>Math Operations>Exponentiate
Main Menu>General Postproc>Surface Operations>Math Operations>Initialize
Main Menu>General Postproc>Surface Operations>Math Operations>Multiply
Main Menu>General Postproc>Surface Operations>Math Operations>Natural Log
Main Menu>General Postproc>Surface Operations>Math Operations>Sine
Main Menu>General Postproc>Surface Operations>Math Operations>Subtract

---

SUCR,*SurfName*,*SurfType*,*nRefine*,*Radius*,*blank*,*blank*,*TolOut*

Create a surface.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SurfName

Eight character surface name.

SurfType

Surface type.

CPLANE

Surface is defined by the cutting plane in window one (controlled by the working plane (/CPLANE,1), NOT the view settings (/CPLANE,0)).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SPHERE

Surface is defined by a spherical surface centered about the working plane origin.

INFC

Surface is defined by a cylindrical surface centered about the working plane origin and extending indefinitely in the positive and negative Z directions.

nRefine

Refinement level.

For SurfType = CPLANE

The refinement level of the surface "mesh". This will be an integer between 0 and 3 (default = 0). See Notes below.

For SurfType = SPHERE

The number of divisions along a 90° arc (minimum = 9). The default is 9.

For SurfType = INFC

The number of divisions along a 90° arc (minimum = 9). The default is 9.

Radius

Appropriate radius value (for INFC or SPHERE).

TolOut

Tolerance value for inclusion of element facets within a prescribed volume. (for INFC)

Notes

This command creates a new surface and stores the following data for that surface:

GCX, GCY, GCZ - global Cartesian coordinates at each point on the surface.

NORMX, NORMY, NORMZ - components of the unit normal at each point on the surface.

DA - the contributory area of each point.

For *SurfType* = CPLANE, *nRefine* refers to the number of points that define the surface. An *nRefine* value of zero is used for points where the element face intersects the cutting plane.

If *SurfType* = CPLANE and *nRefine* = 0, the points reside at the section cuts where the element intersects the cutting plane. Increasing *nRefine* from 0 to 1 will subdivide each surface facet into 4 subfacets, and increase the number of points at which results can be interpolated.

For *SurfType* = CPLANE, the setting from the **/EFACET (p. 602)** command will affect the creation of surface facets and the quality of the fit of the surface in the model. **SUCR (p. 1875)** employs geometry data from PowerGraphics to aid in determining where the surface intersects the model. If **/EFACET (p. 602)**,1 is in effect when the **SUCR (p. 1875)** command is issued, then the curvature of high order

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elements (that is, elements with midside nodes) will be ignored. If your model contains high order elements, you can see a better fit for your surface if /EFACET,2 is in effect when the **SUCR (p. 1875)** command is issued. Currently, the **SUCR (p. 1875)** command interprets **/EFACET (p. 602)**,4 to mean **/EFACET (p. 602)**,2.

For *SurfType* = INFC, a default tolerance of 0.01 will be applied to include the vertices of any facets that fall out of the cylinder definition. This tolerance increases the facet size by one percent to check for inclusion. Excluding facets under such a small tolerance may yield unacceptable (aesthetically) results. Increasing the tolerance by a larger amount (0.1 or 10%) will sometimes yield smother edges along the surface you create.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Inf. Cylinder

Main Menu>General Postproc>Surface Operations>Create Surface>On Cuttng Plane

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>By Dimensions

---

SUDEL, *SurfName*

Delete geometry information as well as any mapped results for specified surface.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SurfName

Eight character surface name.

SurfName = ALL will delete all surface geometry and result infromation.

Menu Paths

Main Menu>General Postproc>Surface Operations>Delete Surfaces

---

SUEVAL, *Parm*, *lab1*, *Oper*

Perform operations on a mapped item and store result in a scalar parameter.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Parm

APDL parameter name.

lab1

Eight character set name for the first set used in calculation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Oper

Operation to perform:

SUM

Sum of *lab1* result values.

INTG

Integral of *lab1* over surface.

AVG

Area-weighted average of a result item [Σ(*lab1**DA) / Σ(DA)]

Notes

The result of this operation is a scalar APDL parameter value. If multiple surfaces are selected when this command is issued, then the operation is carried out on each surface individually and the parameter reperesents the culmulative value of the operation on all selected surfaces.

Menu Paths

Main Menu>General Postproc>Surface Operations>Math Operations>Average Result

Main Menu>General Postproc>Surface Operations>Math Operations>Integrate Results

Main Menu>General Postproc>Surface Operations>Math Operations>Sum of Results

---

SUGET,*SurfName*,*RSetName*,*Parm*,*Geom*

Moves surface geometry and mapped results to an array parameter.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SurfName

Eight character surface name.

RSetName

Eight character result name.

Parm

APDL array parameter name (up to 32 characters).

Geom

Switch controlling how data is written.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON (or 1 or YES)

Writes geometry data and interpolated results information to the parameter.

OFF (or 0 or NO)

Writes only interpolated results information to the parameter. (Default)

Notes

For *Geom* = OFF (or 0 or NO), only results information is written to this parameter.

For *Geom* = ON (or 1 or YES), both geometry data and results information are written to this parameter. Geometry data includes 7 data items: (GCX, GCY, GCZ, NORMX, NORMY, NORMZ, and DA). Results information is then written to the 8th column of the parameter. SetNames of GCX, GCY, GCZ, NORMX, NORMY, NORMZ, and DA are predefined and computed when **SUCR (p. 1875)** is issued.

Menu Paths

Main Menu>General Postproc>Surface Operations>Results to Array

---

SUMAP,*RSetName*,*Item*,*Comp*

Map results onto selected surface(s).

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RSetName

Eight-character name for the result being mapped.

Item

Label identifying the item.

Valid item labels are defined via **PLNSOL (p. 1425)**. Some items also require a component label.

If *Item* = CLEAR, the specified result set is deleted from all selected surfaces

Comp

Component label of item (if required).

Notes

The **SUMAP (p. 1879)** command maps results in the current coordinate system (**RSYS (p. 1639)**) using the selected set of elements.

The command interpolates and stores the results data on to each of the selected surfaces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SUMAP (p. 1879),ALL,CLEAR deletes all results sets from all selected surfaces.

Menu Paths

Main Menu>General Postproc>Surface Operations>Clear Results

SUMTYPE, *Label*

Sets the type of summation to be used in the following load case operations.

POST1 (p. 51): Results (p. 51)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Summation type

COMP

Combine element component stresses only. Stresses such as average nodal stresses, principal stresses, equivalent stresses, and stress intensities are derived from the combined element component stresses. Default.

PRIN

Combine principal stress, equivalent stress, and stress intensity directly as stored on the results file. Component stresses are not available with this option.

Notes

Issue **SUMTYPE (p. 1880)**,PRIN when you want to have a load case operation (**LCOPER (p. 1026)**) act on the principal / equivalent stresses instead of the component stresses. Also issue **SUMTYPE (p. 1880)**,PRIN when you want to read in load cases (**LCASE (p. 1019)**). Note that the **SUMTYPE (p. 1880)** setting is not maintained between /POST1 sessions.

SUMTYPE (p. 1880),PRIN also causes principal nodal values to be the average of the contibuting principal element nodal values (see **AVPRIN (p. 210)**,1).

BEAM188 and BEAM189 elements compute principal stress, equivalent stress, and stress intensity values on request instead of storing them on the results file; **SUMTYPE (p. 1880)**,PRIN does not apply for these elements.

Menu Paths

Main Menu>General Postproc>Load Case>Calc Options>Stress Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SUPL,*SurfName*,*RSetName*,*KWIRE*

Plot result data on all selected surfaces or on a specified surface.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SurfName

Eight-character surface name. *ALL* plots all selected surfaces.

RSetName

Eight-character result name.

KWIRE

Plot in context of model.

0

Plot results without the outline of selected elements.

1

Plot results with the outline of selected elements.

Notes

If *RSetName* is not specified, the surface geometry is plotted. If the Setname portion of the argument is a vector prefix (that is, if result sets of name SetNameX, SetNameY, and SetNameZ exist), Mechanical APDL plots the vectors on the surface as arrows. For example, **SUPL (p. 1881)**,ALL,NORM plots the surface normals as vectors on all selected surfaces, as NORMX, NORMY, and NORMZ are predefined geometry items.

Menu Paths

Main Menu>General Postproc>Surface Operations>Plot Results

Main Menu>General Postproc>Surface Operations>Plot Vectors

SUPR,*SurfName*,*RSetName*

Print global status, geometry information and/or result information.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SurfName

Eight character surface name. If *SurfName* = ALL, repeat printout for all selected surfaces.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RSetName

Eight character result set name.

Notes

When no arguments are specified, **SUPR (p. 1881)** generates a global status summary of all defined surfaces. If only *SurfName* is specified, the geometry information for that surface is printed. If both *SurfName* and *RSetName* are specified, the value of the results set at each point, in addition to the geometry information, is printed.

Menu Paths

Main Menu>General Postproc>Surface Operations>Print Results

Main Menu>General Postproc>Surface Operations>Status>Global

SURESU, --,*Fname*,*Fext*,*Fdir*

Read a set of surface definitions and result items from a file and make them the current set.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

--

Unused field.

Fname

Eight character name.

Fext

Extension name.

Fdir

Optional path specification.

Notes

Reading (and therefore resuming) surface and result definitions from a file overwritea any existing surface definitions.

Reading surfaces back into the postprocessor (**/POST1 (p. 1461)**) does *not* insure that the surfaces (and their results) are appropriate for the model currently residing in **/POST1 (p. 1461)**.

Menu Paths

Main Menu>General Postproc>Surface Operations>Resume Surfaces

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SUSAVE, *Lab*, *Fname*, *Fext*, *Fdir*

Saves surface definitions to a file.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Eight-character surface name.

If *Lab* = ALL (default), then all surfaces are saved to the file.

If *Lab* = S, only currently selected surfaces are saved to the file.

Fname

File name and directory path (248 character maximum, including directory). If you do not specify a directory path, the default is your working directory and you can use all 248 characters for the file name. The file name defaults to the jobname.

Fext

File name extension (eight-character maximum). The extension defaults to "surf".

Fdir

Optional path specification.

Notes

The **SUSAVE (p. 1883)** command saves surface definitions (geometry information)--and any result items mapped onto the surfaces--to a file.

Issuing the **SUSAVE (p. 1883)** command has no effect on the database. The database remains unchanged.

Subsequent executions of the **SUSAVE (p. 1883)** command overwrite previous data in the file.

To read the contents of the file created via the **SUSAVE (p. 1883)** command, issue the **SURESU (p. 1882)** command.

Menu Paths

Main Menu>General Postproc>Surface Operations>Save Surfaces

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SUSEL, *Type*, *Name1*, *Name2*, *Name3*, *Name4*, *Name5*, *Name6*, *Name7*, *Name8*

Selects a subset of surfaces

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of select:

S

Selects a new set (default).

R

Reselects a set from the current set.

A

Additionally selects a set and extends the current set.

U

Unselects a set from the current set.

ALL

Also selects all surfaces.

NONE

Unselects all surfaces.

Name1*, *Name2*, *Name3*, . . . , *Name8

Eight character surface names

Notes

The selected set of surfaces is used in the following operations: **SUMAP (p. 1879)**, **SUDEL (p. 1877)**, **SUCALC (p. 1873)**, **SUEVAL (p. 1877)**, and **SUVECT (p. 1885)**.

Menu Paths

Main Menu>General Postproc>Surface Operations>Select Surfaces

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SUVECT, *RSetName*, *lab1*, *Oper*, *lab2*, *Offset*

Create new result data by operating on two existing result vectors on a given surface.

POST1 (p. 51): Surface Operations (p. 54)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RSetName

Eight character name of the result data output. There will be one or three *RSetName* values depending on the operation specified in *Oper*.

lab1

Eight character name of the mapped data that forms vector 1. Specified sets must exist on all selected surfaces for this operation to take place. The names NORM and GC will be reserved for normals and for global (x, y, z).

Oper

DOT

Computes dot product between *lab1* and *lab2* vectors. The result is a scalar parameter (*RSetName*) and each value within the set can be modified (incremented) via *Offset*.

CROSS

Computes cross product between *lab1* and *lab2* vectors. Each X, Y, Z value in the result can be modified (incremented) via *Offset*.

SMULT

Scales (lab1x, lab1y, lab1z) vector by scalar *lab2*. Each X,Y,Z value in the result can be modified (incremented) via *Offset*.

lab2

Eight character name of the mapped data that forms vector 2. Sets with names Lab2X, Lab2Y, and Lab2Z must exist on all selected surfaces for operation to take place. For *Oper* = SMULT a scalar value or another predefined scalar item (for example, DA) can be supplied.

Offset

An offset value to be applied to the resultant *RSetName*. One value is specified for *Oper* = DOT, and three values are specified for *Oper* = SMULT.

Menu Paths

Main Menu>General Postproc>Surface Operations>Math Operations>Cross Product

Main Menu>General Postproc>Surface Operations>Math Operations>Dot Product

Main Menu>General Postproc>Surface Operations>Math Operations>Scale Result

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SV,*DAMP*,*SV1*,*SV2*,*SV3*,*SV4*,*SV5*,*SV6*,*SV7*,*SV8*,*SV9*

Defines spectrum values to be associated with frequency points.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

DAMP

Damping ratio for this response spectrum curve. If the same as a previously defined curve, the SV values are added to the previous curve. Up to four different curves may be defined, each with a different damping ratio. Damping values must be input in ascending order.

SV1, SV2, SV3, . . . , SV9

Spectrum values corresponding to the frequency points (**FREQ (p. 776)**). Values are interpreted as defined with the **SVTYP (p. 1887)** command. SV values should not be zero. Values required outside the frequency range use the extreme input values.

Notes

Defines the spectrum values to be associated with the previously defined frequency points (**FREQ (p. 776)**). Applies only to the single-point response spectrum. Damping has no effect on the frequency solution. Damping values are used only to identify SV curves for the mode combinations calculation. Only the curve with the lowest damping value is used in the initial mode coefficient calculation. Use **STAT (p. 1861)** command to list current spectrum curve values.

Repeat **SV (p. 1886)** command for additional SV points (100 maximum per *DAMP* curve). SV values are added to the *DAMP* curve after the last nonzero SV value.

The interpolation method between response spectrum points and curves is specified using *KeyInterp* in the **SVTYP (p. 1887)** command. It is logarithmic by default.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Spectr Values

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Spectr Values

SVPLOT,*OptionScale*,*damp1*,*damp2*,*damp3*,*damp4*

Displays input spectrum curves.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

OptionScale

Flag to activate or deactivate input spectrum value scaling:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Do not scale the input spectrum values with scale factor FACT (**SVTYP (p. 1887)** command). This is the default value.

ON

Scale the input spectrum values with scale factor FACT (**SVTYP (p. 1887)** command)

damp1

Damping ratio corresponding to DAMP (**SV (p. 1886)** command) defining the first spectrum curve.

damp2

Damping ratio corresponding to DAMP (**SV (p. 1886)** command) defining the second spectrum curve.

damp3

Damping ratio corresponding to DAMP (**SV (p. 1886)** command) defining the third spectrum curve.

damp4

Damping ratio corresponding to DAMP (**SV (p. 1886)** command) defining the fourth spectrum curve.

Notes

You can display up to four input spectrum tables (**SV (p. 1886)** and **FREQ (p. 776)** commands) with log X scale. If no damping ratio is specified, all spectrum tables are displayed.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

SVTYP,*KSV*,*FACT*,*KeyInterp*

Defines the type of single-point response spectrum.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KSV

Response spectrum type:

0

Seismic velocity response spectrum loading (SV values interpreted as velocities with units of length/time).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Force response spectrum loading (SV values interpreted as force amplitude multipliers).

2

Seismic acceleration response spectrum loading (SV values interpreted as accelerations with units of length/time^2).

3

Seismic displacement response spectrum loading (SV values interpreted as displacements with units of length).

FACT

Scale factor applied to spectrum values (defaults to 1.0). Values are scaled when the solution is initiated (**SOLVE (p. 1822)**). Database values remain the same.

KeyInterp

Key to activate or deactivate the linear interpolation between input response spectrum points and input response spectrum curves:

0 (OFF *or* NO)

Deactivate linear and use logarithmic interpolation. This value is the default.

1 (ON *or* YES)

Activate linear interpolation.

Command Default

Seismic velocity response spectrum.

Notes

Defines the type of single-point response spectrum (**SPOPT (p. 1837)**). The seismic excitation direction is defined with the **SED (p. 1704)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>SinglePt>Settings

Main Menu>Solution>Load Step Opts>Spectrum>SinglePt>Settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SWADD, *Ecomp*, *SHRD*, *NCM1*, *NCM2*, *NCM3*, *NCM4*, *NCM5*, *NCM6*, *NCM7*, *NCM8*, *NCM9*

Adds more surfaces to an existing spot weld set.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ecomp

Name of an existing spot weld set that was previously defined using **SWGEN (p. 1890)**.

SHRD

Search radius. Defaults to 4 times the spot weld radius defined for the spot weld set (*SWRD* on **SWGEN (p. 1890)**).

NCM1, NCM2, NCM3, . . . , NCM9

Surfaces to be added to the spot weld set. Each surface can be input as a predefined node component or a meshed area number.

Notes

This command adds surfaces to an existing spot weld set defined by the **SWGEN (p. 1890)** command. You can add additional surfaces by repeating the **SWADD (p. 1889)** command. However, the maximum number of allowable surfaces (including the 2 surfaces used for the original set defined by **SWGEN (p. 1890)**) for each spot weld set is 11. See Adding Surfaces to a Basic Set for more information.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Add More Surfaces>By Areas

SWDEL, *Ecomp*

Deletes spot weld sets.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ecomp

Name of an existing spot weld set that was previously defined using **SWGEN (p. 1890)**. If *Ecomp* = ALL (default) all spot welds are deleted.

Notes

This command deletes spot weld sets previously defined by the **SWGEN (p. 1890)** command.

SWDEL (p. 1889) does not support deleting spot welds that have the node-to-surface configuration.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Delete

---

SWGEN,*Ecomp*,*SWRD*,*NCM1*,*NCM2*,*SND1*,*SND2*,*SHRD*,*DIRX*,*DIRY*,*DIRZ*,*ITTY*,*ICTY*

Creates a new spot weld set.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ecomp

Name to identify the new spot weld, used for the element component containing the new contact, target, and beam elements generated for the spot weld set.

START

Initiate the spot weld generation process.

END

Finish the spot weld generation process.

When using *Ecomp* to define the component name of the spot weld element, do not begin the name with *STAR* or *END* . Those values are reserved for starting and finishing a block of multiple **SWGEN (p. 1890)** commands.

SWRD

Spot weld radius.

NCM1

Name of a component containing nodes on the first spot weld surface, a meshed area number for the surface, or the name of a component group containing nodal components of multiple surfaces to be weld.

NCM2

Name of a component containing nodes on the second spot weld surface, or a meshed area number for the surface.

SND1

Node number of the first spot weld node corresponding to the first surface (*NCM1*). This node can be on or close to the first surface. For this use of *SND1*, you must also input a node number for *SND2*.

To define multiple spot welds between two surfaces, use the *SND1* field in one of these ways:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To input the name of a table array parameter containing the node information for each spot weld. The node information should be presented as node numbers, geometric coordinates, or both. The table name must be enclosed in "%" signs (for example, %*tabname*%).
- To input the name of a nodal component group of predefined nodes. This use is only supported for node-to-surface configuration of spot welds.
- To input the name of an element component group of predefined joints or beam elements. This use is only supported for node-to-surface configuration of spot welds.

When generating multiple spot welds, *SND2* is ignored.

SND2

Node number of the second spot weld node corresponding to the second surface (*NCM2*). This node can be on or close to the second surface. Mechanical APDL creates the node if not specified.

SHRD

Search radius. Defaults to 4 times the spot weld radius *SWRD*.

DIRX*, *DIRY*, *DIRZ

Spot weld projection direction in terms of normal X, Y, and Z components.

ITTY

Target element type ID.

ICTY

Contact element type ID.

NTOS

Generate the node-to-surface configuration of the spot welds with force-distributed constraints.

NTS2

Generate the node-to-surface configuration of the spot welds with rigid surface constraints.

Notes

This command creates a new spot weld set. You can add more surfaces to the set using **SWADD (p. 1889)** after the initial **SWGEN (p. 1890)** command. However, the maximum number of allowable surfaces (including the two surfaces used for the original set) for each spot weld set is 11.

Ecomp, *SWRD*, *NCM1*, *NCM2*, and *SND1* must be specified. *SND2*, *SHRD*, *DIRX*, *DIRY*, *DIRZ*, *ITTY*, *ICTY* are optional inputs. If the second spot weld node (*SND2*) is specified, that node is used to determine the spot weld projection direction, and *DIRX*, *DIRY* and *DIRZ* are ignored.

To reduce the spot weld generation time of large assembly models where multiple **SWGEN (p. 1890)** commands are issued, define the multiple **SWGEN (p. 1890)** commands within a block initiated with **SWGEN (p. 1890)**,*START* and finished with **SWGEN (p. 1890)**,*END*. The block must be ended with **SWGEN (p. 1890)**,*END* if it is started with **SWGEN (p. 1890)**,*START*. Do not use any commands other than

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SWGEN (p. 1890) and **CMGRP (p. 365)** within the block. Select all elements and nodes, and create all components prior to the **SWGEN (p. 1890)**,*START* command.

If *NCM1* is defined as a component group name to add multiple surfaces to the node-to-surface configuration of the spot weld, no **SWADD (p. 1889)** command is needed. *NCM2* is ignored. The component group of multiple surfaces must be defined using the **CMGRP (p. 365)** command prior to the **SWGEN (p. 1890)**. The maximum allowed number of surfaces for each spot weld set is 8. Using **SWGEN (p. 1890)** to generate multiple spot welds is only supported for node-to-surface configuration of spot welds

If *ITTY* (target element ID) is specified, the following corresponding target element key option must be set: KEYOPT(2) = 1.

If *ICTY* (contact element ID) is specified (only valid for pilot node configuration), the following corresponding contact element key options must be set: KEYOPT(2) = 2, KEYOPT(4) = 1, KEYOPT(12) = 5.

If *ICTY* (contact element ID) is specified as **NTOS** or **NTS2**, node-to-surface configuration of spot-welds is defined. In this configuration, each spot weld consists of a beam element and two node-to-surface MPC contact pairs. Each contact pair has contact element (CONTA175) which is defined by the associated spot-weld node. The target elements (TARGE170) are formed by a group of surface nodes lying within the region of the search radius on the spot weld surfaces. The program creates real constant IDs for each layer, and proper contact and target element type IDs. The following corresponding contact element key options are set: KEYOPT(2) = 2, KEYOPT(12) = 5. For*ICTY*=**NTOS**, target elements KEYOPT(5) = 4 is set for defining the force-distributed constraint type. For *ICTY*=**NTS2**, target elements KEYOPT(5) = 7 is set for defining the rigid surface constraint type.

After *SND1* and *SND2* are projected onto surface 1 and surface 2, respectively, two new pilot-nodes (which represent the spot weld nodes) are generated at the locations of *SND1* and *SND2* and meshed with TARGE170 target elements (**TSHAP (p. 2023)**,PILO).

By default, the contact pair created at each spot weld surface is an MPC-based force-distributed constraint. To use force-distributed constraints based on the Lagrange multiplier method, you must set KEYOPT(2) = 3 for the contact elements after **SWGEN (p. 1890)** is issued. To use rigid constraints instead of force-distributed constraints, you must set KEYOPT(4) = 0 for the contact elements after **SWGEN (p. 1890)** is issued.

To automatically define multiple spot welds between two surfaces, use tabular input in the *SND1* field. The table array that you input must be a 2D array parameter. For more information, see Generation of Multiple Spot Welds in the *Contact Technology Guide*.

To use the relaxation method to eliminate overconstraint, you must set KEYOPT(11) = 1 for the target elements after **SWGEN (p. 1890)** is issued.

Issue **SWLIST (p. 1893)** and **SWDEL (p. 1889)** to list or delete spot welds, respectively. **SWLIST (p. 1893)** and **SWDEL (p. 1889)** are not supported for node-to-surface configuration. For more information about defining spot welds, see Creating a Basic Spot Weld Set with SWGEN.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Add More Surfaces>By Node Component

Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Create New Set>By Area

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>Create New Set>By Node Component

---

SWLIST, *Ecomp*

Lists spot weld sets.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ecomp

Name of an existing spot weld set that was previously defined using **SWGEN (p. 1890)**. If *Ecomp* = ALL (default), all spot weld sets are listed.

Notes

This command lists spot weld node, beam, and contact pair information for all defined spot weld sets, or for the specified set. To ensure that all defined spotwelds are listed, issue **CMSEL (p. 374)**,ALL (to select all components) before issuing the **SWLIST (p. 1893)** command.

When **SWLIST (p. 1893)** is issued in POST1, the beam forces and moments are output. For the case of a deformable spot weld, the stresses are also output in the beam local coordinate system.

SWLIST (p. 1893) does not support listing spot welds that have the node-to-surface (NTOS) configuration.

Menu Paths

Main Menu>General Postproc>List Results>SpotWeld Solution

Main Menu>Preprocessor>Modeling>Create>Elements>SpotWeld>List

Utility Menu>List>Results>SpotWeld Solution

SYNCHRO, *RATIO*, *Cname*

Specifies whether the excitation frequency is synchronous or asynchronous with the rotational velocity of a structure.

SOLUTION (p. 38): Inertia (p. 45)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RATIO

In a stationary reference frame (**CORIOLIS (p. 417)** with *RefFrame* = ON), *RATIO* is the ratio between the frequency of excitation and the frequency of the rotational velocity of the structure. This value must be greater than 0. The default is an unbalance excitation.

In a rotating reference frame (**CORIOLIS (p. 417)** with *RefFrame* = OFF), *RATIO* is the ratio between the frequency of excitation and the frequency of the rotational velocity of the structure minus 1. This value must be greater than 0. There is no default.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cname

The name of the rotating component on which to apply the harmonic excitation.

Notes

The **SYNCHRO (p. 1893)** command specifies whether the excitation frequency is synchronous or asynchronous with the rotational velocity of a structure in a harmonic analysis. Use the command to take into account rotating harmonic forces on rotating structures.

Mechanical APDL calculates the rotational velocity Ω of the structure from the excitation frequency *f*, defined (via the **HARFRQ (p. 885)** command) as $\Omega = 2\pi f$ / *RATIO*. The rotational velocity is applied along the direction cosines of the rotation axis (specified via an **OMEGA (p. 1324)** or **CMOMEGA (p. 368)** command).

In a stationary reference frame, specifying any value for *RATIO* causes a general rotational force excitation and not an unbalance force. To define an unbalance excitation force (F = Ω^2 * Unb), *RATIO* should be left blank (the nodal unbalance Unb is specified via the **F (p. 729)** command).

In a rotating reference frame (**CORIOLIS (p. 417)** with *RefFrame* = OFF), an unbalance excitation is a static load; therefore, a value must be supplied for *RATIO*.

The **SYNCHRO (p. 1893)** command is valid only for the full harmonic analysis method (**HROPT (p. 912)**,FULL) and the frequency-sweep harmonic analysis method (**HROPT (p. 912)**,VT) involving a rotating structure (**OMEGA (p. 1324)** or **CMOMEGA (p. 368)**) with Coriolis enabled (**CORIOLIS (p. 417)**).

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Inertia>Angular Veloc>Synchronous

Main Menu>Solution>Define Loads>Apply>Structural>Inertia>Angular Veloc>Synchronous

---

/SYP,*String*,*ARG1*,*ARG2*,*ARG3*,*ARG4*,*ARG5*,*ARG6*,*ARG7*,*ARG8*

Passes a command string and arguments to the operating system.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

String

Command string (cannot include commas). See also the **/SYS (p. 1895)** command.

ARG1, ARG2, ARG3, . . . , ARG8

Arguments to be appended to the command string, separated by blanks, commas, or other delimiter characters (see the *Operations Guide*). The arguments may be numbers, parameters, or parametric expressions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Passes a command string to the operating system for execution, along with arguments to be appended to the command string. See the *Operations Guide* for details. Mechanical APDL may not be aware of your specific user environment. For example, on Linux this command may not recognize aliases, depending on the hardware platform and user environment.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

/SYS,*String*

Passes a command string to the operating system.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

String

Command string, up to 639 characters (including blanks, commas, etc.). The specified string is passed verbatim to the operating system, that is, no parameter substitution is performed.

Notes

Passes a command string to the operating system for execution (see the *Operations Guide*). Typical strings are system commands such as list, copy, rename, etc. Control returns to Mechanical APDL after the system procedure is completed. Mechanical APDL may not be aware of your specific user environment. For example, on Linux this command may not recognize aliases, depending on the hardware platform and user environment.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

T Commands

TALLOW, *TEMP1*, *TEMP2*, *TEMP3*, *TEMP4*, *TEMP5*, *TEMP6*

Defines the temperature table for safety factor calculations.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TEMP1, TEMP2, TEMP3, . . . , TEMP6

Input up to six temperatures covering the range of nodal temperatures. Temperatures must be input in ascending order.

Notes

Defines the temperature table for safety factor calculations (**SFACT (p. 1743)**, **SALLOW (p. 1644)**). Use **STAT (p. 1861)** command to list current temperature table. Repeat **TALLOW (p. 1897)** command to zero table and redefine points (6 maximum).

Safety factor calculations are not supported by PowerGraphics. Both the **SALLOW (p. 1644)** and **TALLOW (p. 1897)** commands must be used with the Full Model Graphics display method active.

Menu Paths

Main Menu>General Postproc>Safety Factor>Allowable Strs>Reset Temps

Main Menu>General Postproc>Safety Factor>Allowable Strs>Temp-depend

TARGET, *Nlist*

Specifies the target nodes for mapping pressures onto surface effect elements.

/MAP (p. 63): Pressure Mapping (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Nlist

Nodes defining the surface upon which the pressures will be mapped. Use the label ALL or specify a nodal component name. If ALL, all selected nodes (**NSEL (p. 1266)**) are used (default). Individual nodes may not be entered.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The node list specified by *Nlist* must contain a sufficient number of nodes to define an element surface. The surface must be meshed (**ESURF (p. 682)** command) with SURF154 elements prior to issuing this command.

Menu Paths

This command cannot be accessed from a menu.

***TAXIS**,*ParmLoc*,*nAxis*,*Val1*,*Val2*,*Val3*,*Val4*,*Val5*,*Val6*,*Val7*,*Val8*,*Val9*,*Val10*

Defines table index numbers.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParmLoc

Name and starting location in the table array parameter for indexing. Indexing occurs along the axis defined with *nAxis*.

nAxis

Axis along which indexing occurs. Valid labels are:

1 --

Corresponds to Row. Default.

2 --

Corresponds to Column.

3 --

Corresponds to Plane.

4 --

Corresponds to Book.

5 --

Corresponds to Shelf.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL --

Lists all index numbers. Valid only if *Val1* = LIST.

Val1*, *Val2*, *Val3*, ..., *Val10

Values of the index numbers for the axis *nAxis*, starting from the table array parameter location *ParmLoc*. You can define up to ten values.

To list the index values specified with *nAxis*, issue *Val1* = LIST. If *Val1* = LIST, *Val2* - *Val10* are ignored.

Notes

***TAXIS (p. 1898)** is a convenient method to define table index values. These values reside in the zero column, row, etc. Instead of filling values in these zero location spots, use the ***TAXIS (p. 1898)** command. For example,

```
*TAXIS,longtable(1,4,1,1),2,1.0,2.2,3.5,4.7,5.9
```

would fill index values 1.0, 2.2, 3.5, 4.7, and 5.9 in *nAxis* 2 (column location), starting at location 4.

To list index numbers, issue ***TAXIS (p. 1898)**,*ParmLoc*, *nAxis*, LIST, where *nAxis* = 1 through 5 or ALL.

Menu Paths

This command cannot be accessed from a menu.

TB, *Lab*, *MATID*, *NTEMP*, *NPTS*, *TBOPT*, *--*, *FuncName*

Activates a data table for material properties or special element input.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 1955)

For a list of *elements* and the material models they support (*Lab* value), see Element Support for Material Models in the *Element Reference*.

For a list of *material models* and the elements that support them, see Material Model Support for Elements in the *Material Reference*.

Lab

Material model data table type:

AFDM

Acoustic frequency-dependent material. (p. 1905)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AHYPER

Anisotropic hyperelasticity (p. 1906).

ANEL

Anisotropic elasticity (p. 1907).

ANISO

Generalized Hill anisotropy (p. 1908).

AVIS

Anisotropic viscosity (p. 1908).

BB

Bergstrom-Boyce (p. 1908).

BH

Magnetic field (p. 1909).

CAST

Cast iron (p. 1910).

CDM

Damage (p. 1910).

CFOAM

Crushable foam (p. 1912).

CGCR

Crack-growth (p. 1912) fracture criterion (**CGROW (p. 321)**).

CHABOCHE

Chaboche nonlinear kinematic hardening (p. 1913) using von Mises or Hill plasticity.

CONCR

Concrete (p. 1914) element or material data.

CREEP

Creep (p. 1915). Pure creep, creep with isotropic hardening plasticity, or creep with kinematic hardening plasticity using both von Mises or Hill potentials.

CRKI

– Material criterion for adaptive-crack initiation (p. 1916) (**ADPCI (p. 98)**)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CTE

Coefficient of thermal expansion (p. 1916).

CZM

Cohesive zone (p. 1917).

DENS

Mass Density (p. 1919).

DLST

Anisotropic dielectric loss tangent (p. 1919).

DMGE

Damage evolution law (p. 1919).

DMGI

Damage initiation criteria (p. 1920).

DPER

Anisotropic electric permittivity (p. 1920).

EDP

Extended Drucker-Prager (p. 1921) (for granular materials such as rock, concrete, soil, ceramics and other pressure-dependent materials).

ELASTIC

Elasticity (p. 1922).

ELST

Anisotropic elastic loss tangent (p. 1923).

EXPE

Experimental data (p. 1923).

FCON

Fluid conductance data (p. 1924).

FCLI

Material strength limits (p. 1925) for calculating failure criteria.

FLUID

Fluid (p. 1925).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FRIC

Coefficient of friction (p. 1926) based on Coulomb's Law or user-defined friction.

GASKET

Gasket (p. 1927).

GURSON

Gurson pressure-dependent plasticity (p. 1928) for porous metals.

HFLM

Film coefficient data (p. 1928).

HILL

Hill anisotropy (p. 1929). When combined with other material options, simulates plasticity, viscoplasticity, and creep -- all with the Hill potential.

HYPER

Hyperelasticity (p. 1929) material models (Arruda-Boyce, Blatz-Ko, Extended Tube, Gent, Hencky, Mooney-Rivlin [default], Neo-Hookean, Ogden, Ogden Foam, Polynomial Form, Response Function, Yeoh, and user-defined).

INTER

Contact interaction (p. 1931).

JOIN

Joint (p. 1932) (linear and nonlinear elastic stiffness, linear and nonlinear damping, and frictional behavior).

JROCK

Jointed rock (p. 1936).

MC

Mohr-Coulomb (p. 1936).

MELAS

Multilinear elasticity (p. 1937).

MIGR

Migration (p. 1938).

MPLANE

Microplane (p. 1938).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NLISO

Voce isotropic hardening law (p. 1939) (or power law) for modeling nonlinear isotropic hardening using von Mises or Hill plasticity.

PELAS

Porous elasticity (p. 1942).

PERF

Perforated material (p. 1939) for acoustics; equivalent fluid model of perforated media, poroelastic material model, and transfer admittance matrix.

PIEZ

Piezoelectric matrix (p. 1941).

PLASTIC

Nonlinear plasticity (p. 1941).

PM

Porous media (p. 1942). Coupled pore-fluid diffusion and structural model of porous media.

PRONY

Prony series (p. 1943) constants for viscoelastic materials.

PZRS

Piezoresistivity (p. 1944).

RATE

Rate-dependent plasticity (p. 1945) (viscoplasticity) when combined with the BISO, NLISO or PLASTIC material options, or rate-dependent anisotropic plasticity (anisotropic viscoplasticity) when combined with the HILL and BISO, NLISO or PLASTIC material options.

The exponential visco-hardening option includes an explicit function for directly defining static yield stresses of materials.

The Anand unified plasticity option requires no combination with other material models.

RO

Ramberg-Osgood (p. 1945).

SDAMP

Material damping coefficients (p. 1946).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SHIFT

Shift function (p. 1946) for viscoelastic materials.

SINT

Sintering (p. 1947). Available with the Additive Suite license.

SMA

Shape memory alloy (p. 1948) for simulating superelasticity, shape memory effect, or shape memory effect with plasticity.

SOIL

Soil (p. 1949) models.

STATE

User-defined state variables (p. 1949). Valid with **TB (p. 1899)**,USER and used with either the `UserMat` or `UserMatTh` subroutine. Also valid with **TB (p. 1899)**,CREEP when *TBOPT* = 100 (implicit creep) and used with the `UserCreep` subroutine, or when real constant C6 = 100 (explicit creep) and used with the UserCr subroutine.

SWELL

Swelling (p. 1950) strain function.

TNM

Three-network model (p. 1951) for viscoplastic materials.

THERM

Thermal properties (p. 1950).

USER

User-defined (p. 1952) material or thermal material model (general-purpose except for incompressible material models) or thermal material model.

WEAR

Contact surface wear (p. 1953).

XTAL

Crystal plasticity (p. 1954) for elasto-viscoplastic crystalline materials.

MATID

Material reference identification number. Valid value is any number n, where $0 < n < 100{,}000$. Default = 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NTEMP

The number of temperatures for which data will be provided (if applicable). Specify temperatures via the **TBTEMP (p. 1993)** command.

NPTS

For most labels where *NPTS* is defined, the number of data points to be specified for a given temperature. Define data points via the **TBDATA (p. 1960)** or **TBPT (p. 1992)** commands.

- -

Unused field.

FuncName

The name of the function to be used (entered as %*tabname*%, where *tabname* is the name of the table created by the Function Tool). Valid only when *Lab* = JOIN (joint element material) and nonlinear stiffness or damping are specified on the *TBOPT* field (see "JOIN -- Joint Element Specifications" (p. 1932)). The function must be predefined via the Function Tool. To learn more about how to create a function, see Using the Function Tool in the *Basic Analysis Guide*.

Data Table Specifications

Following are input requirements (*NTEMP*, *NPTS*, and *TBOPT* values) and links to detailed documentation for each data table type (**TB (p. 1899)**,*Lab* value):

AFDM -- Acoustic Frequency-Dependent Material Specifications

NTEMP:

Not used.

***NPTS*:**

Not used.

TBOPT:

Acoustic material options:

MAT

Material properties

THIN

Thin layer

RECT

Rectangular cross-section

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CIRC

Circular cross-section

ROOM

Diffusion properties for room acoustics

References:

Defining Acoustic Material Properties in the *Acoustic Analysis Guide*

Acoustic Frequency-Dependent Materials in the *Material Reference*

See **TBFIELD (p. 1966)** for more information about defining temperature- and/or frequency-dependent properties.

AHYPER -- Anisotropic Hyperelasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. Maximum = 40.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

Anisotropic hyperelastic material options.

POLY --

Polynomial strain energy potential.

EXPO --

Exponential strain energy potential.

AVEC --

Define the A vector.

BVEC --

Define the B vector.

PVOL --

Volumetric potential.

USER --

User-defined potential invariant set type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UNUM --

User-defined invariant set number.

AU01 --

User-defined material parameters.

FB01 --

User-defined fiber directions.

References:

Anisotropic Hyperelasticity (**TB**,AHYPER) in the *Material Reference*

Anisotropic Hyperelasticity Model in the *Structural Analysis Guide*

Subroutine UserHyperAniso (Writing Your Own Anisotropic Hyperelasticity Laws) in the *Programmer's Reference*

Anisotropic Hyperelasticity in the *Theory Reference*

ANEL -- Anisotropic Elasticity Specifications

This material model is not supported for use with the coefficient of thermal expansion (p. 1916) (**TB (p. 1899)**,CTE). The maximum number of ANEL tables is 1,000,000.

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 6. Maximum = 6.

***NPTS*:**

Not used.

***TB0PT*:**

Anisotropic elastic matrix options.

0 --

Elasticity matrix used as supplied (input in stiffness form).

1 --

Elasticity matrix inverted before use (input in flexibility form).

References:

Anisotropic Elasticity in the *Material Reference*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ANISO – Generalized Hill Anisotropy Specifications

NTEMP :

Not used.

***NPTS*:**

Not used.

TB0PT :

Not used.

References:

Generalized Hill Anisotropy in the *Material Reference*

AVIS -- Anisotropic Viscosity Specifications

NTEMP :

Not used.

***NPTS*:**

Not used.

TB0PT :

Anisotropic viscosity matrix options:

0

Viscosity matrix (used as specified).

1

Fluency matrix (converted to viscosity matrix before use).

References:

Anisotropic Viscosity in the *Material Reference*

BB -- Bergstrom-Boyce Hyperelasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. The maximum must be a value such that (*NTEMP* x *NPTS*) <= 1000.

***NPTS*:**

Number of material constants.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***TB0PT*:**

Isochoric or volumetric strain-energy function:

ISO --

Define material constants for isochoric strain energy.

PVOL --

Define material constants for volumetric strain energy.

References:

Bergstrom-Boyce in the *Theory Reference*

Bergstrom-Boyce Material (**TB**,BB) in the *Material Reference*

Bergstrom-Boyce Hyperviscoelasticity Model in the *Structural Analysis Guide*

BH -- Magnetic Field Data Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Number of data points to be specified. Default = 20. Maximum = 500.

***TB0PT*:**

BH curve options.

BH or (blank) --

BH curve data (default).

TCF --

Thermal coefficient data for BH curve modification. This option is valid for the following elements: PLANE223, SOLID226, SOLID227, PLANE233, SOLID236, and SOLID237.

References:

Magnetism in the *Material Reference*

Additional Guidelines for Defining Regional Material Properties and Real Constants in the *Low-Frequency Electromagnetic Analysis Guide*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CAST -- Cast Iron Plasticity Specifications

NTEMP :

Number of temperatures for which data will be provided. Default = 1. Maximum = 10.

NPTS :

Not used.

TB0PT :

Cast iron options:

ISOTROPIC --

Specifies cast iron plasticity with isotropic hardening.

TENSION --

Defines stress-strain relation in tension.

COMPRESSION --

Defines stress-strain relation in compression.

ROUNDING --

Defines tension yield surface rounding factor.

References:

Cast Iron in the *Material Reference*

CDM -- Damage Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. The maximum must be a value such that (*NTEMP* x *NPTS*) <= 1000.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

Damage option:

PSE2 --

Mullins effect for hyperelasticity models: pseudo-elastic model with a modified Ogden-Roxburgh damage function. Requires *NPTS* = 3.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MUSER --

Mullins effect for hyperelasticity models: Pseudo-elastic model with a user-defined damage function.

GDMG --

Generalized damage model parameters.

FIB1 --

Damage parameters in fiber direction 1.

FIB2 --

Damage parameters in fiber direction 2.

FIB3 --

Damage parameters in fiber direction 3.

DUCTILE --

Damage-initiation parameters for ductile damage.

EXPDMG --

Exponential damage-evolution parameters for ductile damage.

LINDMG --

Linear damage-evolution parameters for ductile damage.

VREG --

Viscous regularization parameters for ductile damage.

References:

Mullins Effect in the *Theory Reference*

Mullins Effect (**TB**,CDM) in the *Material Reference*

Mullins Effect Model in the *Structural Analysis Guide*

Subroutine userMullins (Writing Your Own Pseudo-Elastic Mullins Effect Law) in the *Programmer's Reference*

Regularized Generalized Damage for Fatigue and Thermomechanical Fatigue in the *Material Reference*

Regularized Anisotropic Damage in the *Material Reference*

Ductile Damage in the *Material Reference*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CFOAM -- Crushable Foam Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

***TB0PT*:**

Crushable foam material option:

YIELD --

Initial yield stress values.

HTYPE --

Hardening evolution type.

MHARD --

Multilinear hardening evolution points.

PPR --

Plastic Poisson's ratio.

References:

Crushable Foam in the *Material Reference*

CGCR -- Crack-Growth Fracture Criterion

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

Fracture-criterion option.

LINEAR -- Linear fracture criterion. Valid when *NPTS* = 3.

BILINEAR -- Bilinear fracture criterion. Valid when *NPTS* = 4.

BK -- B-K fracture criterion. Valid when *NPTS* = 3.

MBK -- Modified B-K (Reeder) fracture criterion. Valid when *NPTS* = 4.

POWERLAW -- Wu's Power Law fracture criterion. Valid when *NPTS* = 6.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

USER -- User-defined fracture criterion. Valid when *NPTS* = 20.

PSMAX -- Circumferential stress criterion based on $\sigma_{r\theta}= 0$ when sweeping around the crack tip at a given radius. Valid when *NPTS* = 1. Used in an XFEM-based crack-growth analysis only.

STTMAX -- Maximum circumferential stress criterion. Valid when *NPTS* = 1. Used in an XFEM-based crack-growth analysis only.

RLIN -- Rigid linear evolution law for the decay of stress. Valid when *NPTS* = 4. Used in an XFEM-based crack-growth analysis only.

PARIS -- Paris' Law for fatigue crack-growth. Valid when *NPTS* = 2. Used in a SMART- *or* XFEM-based fatigue crack-growth analysis only.

WALK -- Walker equation for fatigue crack-growth. Valid when *NPTS* = 3. Used in a SMART-based fatigue crack-growth analysis only.

FORM -- Forman equation for fatigue crack-growth. Valid when *NPTS* = 3. Used in a SMART-based fatigue crack-growth analysis only.

TFDK -- Tabular fatigue law for fatigue crack-growth. Used in a SMART-based fatigue crack-growth analysis only.

NG03 -- NASGRO equation v. 3 for fatigue crack-growth. Valid when *NPTS* = 9. Used in a SMART-based fatigue crack-growth analysis only.

NG04 -- NASGRO equation v. 4 for fatigue crack-growth. Valid when *NPTS* = 10. Used in a SMART-based fatigue crack-growth analysis only.

KIC -- Critical stress-intensity factor for static crack-growth. Valid when *NPTS* = 1. Valid in a SMART-based static crack-growth analysis only.

JIC -- Critical J-integral for static crack-growth. Valid when *NPTS* = 1. Valid in a SMART-based static crack-growth analysis only.

Fatigue crack-closure option. Valid in a SMART-based fatigue crack-growth analysis only, with crack-growth based on Paris' law or tabular fatigue law.

ELBER – Elber closure function.

SCHIJVE – Schijve closure function.

NEWMAN – Newman closure function.

UPOLY – Polynomial closure function.

References:

Fracture Analysis Guide

CGROW (p. 321) command

CHABOCHE -- Chaboche Nonlinear Kinematic Hardening Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. The maximum value of *NTEMP* is such that *NTEMP* x (1 + 2*NPTS*) = 1000.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***NPTS*:**

Number of kinematic models to be superposed. Default = 1. Maximum = 5.

***TB0PT*:**

(blank) --

Default option for nonlinear kinematic hardening.

TRATE --

Include temperature-rate term in back-stress evolution.

SHDR --

Strain-hardening of dynamic recovery properties. To use this option, *TB0PT* = TRATE is also required.

References:

Nonlinear Kinematic Hardening in the *Material Reference*

CONCR -- Concrete Plasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided (used only if *TB0PT* = 0 or 1). Default = 6. Maximum = 6.

***NPTS*:**

Not used.

***TB0PT*:**

Concrete material options.

DP --

Drucker-Prager concrete strength parameters.

RCUT --

Rankine tension failure parameter.

DILA --

Drucker-Prager concrete dilatation.

HSD2 --

Drucker-Prager concrete exponential hardening/softening/dilitation (HSD) behavior.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HSD4 --

Drucker-Prager concrete steel reinforcement HSD behavior.

HSD5 --

Drucker-Prager concrete fracture energy HSD behavior.

HSD6 --

Drucker-Prager concrete linear HSD behavior.

FPLANE --

Drucker-Prager concrete joint parameters.

FTCUT --

Drucker-Prager concrete joint tension cutoff.

FORIE --

Drucker-Prager concrete joint orientation.

MW --

Menetrey-Willam constitutive model.

MSOL --

Material solution option.

References:

Drucker-Prager Concrete in the *Material Reference*

Hardening, Softening and Dilatation (HSD) Behavior in the *Material Reference*

Menetrey-Willam in the *Material Reference*

CREEP -- Creep Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

Creep model options:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Explicit creep option. See Explicit Creep Equations for available equations.

1 through 13 --

Implicit creep option. See Implicit Creep Equations for available equations.

100 --

User-defined creep option (implicit). Define the creep law via the UserCreep subroutine.

References:

Creep Option in the *Material Reference*

Creep Model in the *Structural Analysis Guide*

See also Combining Material Models in the *Material Reference*.

CRKI -- Adaptive Crack-Initiation Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 6. Maximum = 6.

***NPTS*:**

Not used.

***TBOPT*:**

Adaptive crack-initiation options:

PSMAX --

Maximum principal stress (default).

References:

SMART Method for Crack-Initiation Simulation in the *Fracture Analysis Guide*

CTE -- Coefficient of Thermal Expansion Specifications

***NTEMP*:**

No limit.

***NPTS*:**

Not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TB0PT :

(blank) --

Enter the secant coefficients of thermal expansion (CTEX,CTEY,CTEZ) (default).

USER --

User-defined thermal strain.

FLUID --

Fluid thermal-expansion coefficient for porous media.

UFSTRAIN --

User-defined free strain in porous media.

References:

Thermal Expansion in the *Material Reference*

Porous Media Mechanics in the *Material Reference*

Subroutine userthstrain (Defining Your Own Thermal Strain) in the *Programmer's Reference*

Free-Strain Rate in the *Material Reference*

Subroutine userfreestrain (Defining Your Own Free-Strain Increment) in the *Programmer's Reference*

See also **TBFIELD (p. 1966)** (for defining frequency-dependent, temperature-dependent, and user-defined field-variable-based properties).

CZM -- Cohesive Zone Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

Cohesive zone material options.

EXPO --

Exponential material behavior. Valid for interface elements and contact elements.

BILI --

Bilinear material behavior. Valid for interface elements, contact elements, and in an XFEM-based crack-growth analysis when cohesive behavior on the initial crack is desired.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CBDD --

Bilinear material behavior with linear softening characterized by maximum traction and maximum separation. Valid for contact elements only.

CBDE --

Bilinear material behavior with linear softening characterized by maximum traction and critical energy release rate. Valid for contact elements only.

CEXP --

Exponential material behavior for preventing surface penetration on the cohesive interface. Valid for SMART-based crack-growth only.

CLIN --

Linear material behavior with a penalty slope for preventing surface penetration on the cohesive interface. Valid for SMART-based crack-growth only.

FRIC --

Isotropic Coulomb's frictional law on the cohesive interface. Valid for SMART-based crack-growth with *TB0PT* = CLIN or *TB0PT* = CEXP only.

REXP --

Rigid exponential material behavior on the cohesive interface. Valid for SMART-based crack-growth only.

VREG --

Viscous regularization. Valid for interface elements and contact elements. Also valid in an XFEM-based crack-growth analysis when cohesive behavior is specified for the initial crack.

USER --

User-defined option. Valid for interface elements only.

References:

Cohesive Zone Material (CZM) Model in the *Theory Reference*

Cohesive Material Law in the *Material Reference*

Subroutine userCZM (Creating Your Own Cohesive Zone Material) in the *Programmer's Reference*

Crack-Initiation and -Growth Simulation, Interface Delamination, and Fatigue Crack-Growth in the *Fracture Analysis Guide*

XFEM-Based Crack Analysis and Crack-Growth Simulation in the *Fracture Analysis Guide*

Enhancing Crack Surfaces with Cohesive Zone Elements in the *Fracture Analysis Guide*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DENS -- Mass Density Specifications

***NTEMP*:**

Not used.

***NPTS*:**

1

***TB0PT*:**

Not used.

References:

See **TBFIELD (p. 1966)** and User-Defined Field Variables in the *Material Reference* for more information about defining temperature-dependent and/or user-defined field-variable-based properties.

DLST -- Anisotropic Dielectric Loss Tangent Specifications

NTEMP :

Not used.

***NPTS*:**

Not used.

TB0PT :

Not used.

References:

Anisotropic Dielectric Loss Tangent in the *Material Reference*

DMGE -- Damage Evolution Law Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 4 when *TB0PT* = MPDG

***TB0PT*:**

Damage initiation definition:

1 *or* MPDG --

Progressive damage evolution based on simple instant material stiffness reduction.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 *or* CDM --

Progressive damage evolution based on continuum damage mechanics.

Reference:

Damage-Evolution Law in the *Material Reference*

DMGI -- Damage Initiation Criteria Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 4 when *TB0PT* = FCRT.

***TB0PT*:**

Damage initiation definition:

1 or FCRT --

Define failure criteria as the damage initiation criteria.

Reference:

Damage Initiation Criteria

DPER -- Anisotropic Electric Permittivity Specifications

NTEMP :

Not used.

NPTS :

Not used.

TB0PT :

Permittivity matrix options for PLANE222, PLANE223, SOLID225, SOLID226, and SOLID227:

0 --

Permittivity matrix at constant strain $[\varepsilon^S]$ (used as supplied)

1 --

Permittivity matrix at constant stress $[\varepsilon^T]$ (converted to $[\varepsilon^S]$ form before use)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References:

Anisotropic Electric Permittivity in the *Material Reference*

EDP -- Extended Drucker-Prager Plasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. Maximum = 40.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

EDP material options.

LYFUN --

Linear yield function.

PYFUN --

Power law yield function.

HYFUN --

Hyperbolic yield function.

LFPOT --

Linear flow potential function.

PFPOT --

Power law flow potential function.

HFPOT --

Hyperbolic flow potential function.

CYFUN --

Cap yield function.

CFPOT --

Cap flow potential function.

References:

Extended Drucker-Prager (EDP) in the *Material Reference*

Extended Drucker-Prager Cap in the *Material Reference*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ELASTIC -- Elasticity Specifications

NTEMP :

Number of temperatures for which data will be provided.

NPTS :

Number of properties to be defined for the material option. This value is set automatically according to the elasticity option (*TB0PT*) selected. If *TB0PT* is not specified, default settings become *NPTS* = 2 and *TB0PT* = ISOT.

TB0PT :

Elasticity options:

ISOT --

Isotropic property (EX, NUXY) (default). Setting *NPTS* = 2 also selects this option automatically.

OELN --

Orthotropic option with minor Poisson's ratio (EX, EY, EZ, GXY, GYZ, GXZ, NUXY, NUYZ, NUXZ). *NPTS* = 9. Setting *NPTS* = 9 selects this option automatically. All nine parameters must be set, even for the 2D case.

OELM --

Orthotropic option with major Poisson's ratio (EX, EY, EZ, GXY, GYZ, GXZ, PRXY, PRYZ, PRXZ). *NPTS* = 9. All nine parameters must be set, even for the 2D case.

AELS --

Anisotropic option in stiffness form (D11, D21, D31, D41, D51, D61, D22, D32, D42, D52, D62, D33, D43, ... D66). *NPTS* = 21. Setting *NPTS* = 21 selects this option automatically.

AELF --

Anisotropic option in compliance form (C11, C21, C31, C41, C51, C61, C22, C32, C42, C52, C62, C33, C43, ... C66). *NPTS* = 21.

FIB1 --

Fiber parameters in fiber direction 1.

FIB2 --

Fiber parameters in fiber direction 2.

FIB3 --

Fiber parameters in fiber direction 3.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

USER --

User-defined linear elastic properties. For more information on the `user_tbelastic` subroutine, see the Guide to User-Programmable Features in the *Programmer's Reference*.

References:

See **TBFIELD (p. 1966)** for more information about defining temperature- and/or frequency-dependent properties.

Regularized Anisotropic Damage Response in the *Material Reference*

Full Harmonic Analysis in the *Structural Analysis Guide*

ELST -- Anisotropic Elastic Loss Tangent Specifications

NTEMP :

Not used.

***NPTS*:**

Not used.

TB0PT :

Not used.

References:

Anisotropic Elastic Loss Tangent in the *Material Reference*

EXPE -- Experimental Data Specifications

***NTEMP*:**

Number of temperatures for which data will be provided.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

Experimental data type:

UNITENSION --

Uniaxial tension experimental data.

UNICOMPRESSION --

Uniaxial compression experimental data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UNIAXIAL --

Uniaxial experimental data (combined uniaxial tension and compression).

BIAXIAL --

Equibiaxial experimental data.

SHEAR --

Pure shear experimental data (also known as planar tension).

SSHEAR --

Simple shear experimental data.

VOLUME --

Volumetric experimental data.

GMODULUS --

Shear modulus experimental data.

KMODULUS --

Bulk modulus experimental data.

EMODULUS --

Tensile modulus experimental data.

NUXY --

Poisson's ratio experimental data.

References:

Experimental Data in the *Material Reference*

Experimental Response Functions in the *Theory Reference*

Viscoelasticity in the *Material Reference*

See also **TBFIELD (p. 1966)** for information about defining field-dependent experimental data.

FCON -- Fluid Conductance Data Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. Maximum = 20.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 1. Maximum = 100.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***TB0PT*:**

Not used.

References:

FLUID116

FCLI -- Material Strength Limits Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 20 when *TB0PT* = 1. Default = 9 when *TB0PT* = 2.

***TB0PT*:**

Material strength limit definition:

1 --

Define stress-strength limits.

2 --

Define strain-strength limits.

References:

Material Strength Limits in the *Material Reference*

FLUID -- Fluid Specifications

NTEMP :

Number of temperatures for which data will be provided. Default = 1.

NPTS :

Number of data points to be specified for a given temperature.

TB0PT :

Fluid material options:

LIQUID --

Define material constants for a liquid material.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GAS --

Define material constants for a gas material.

PVDATA --

Define pressure-volume data for a fluid material.

References:

Fluids in the *Material Reference*

Fluid Material Models in the *Theory Reference*

FRIC -- Coefficient of Friction Specifications

NTEMP :

Number of temperatures for which data will be provided. Default = 1. No maximum limit.

NTEMP is not used for the following situations:

- Isotropic or orthotropic friction defined in terms of field data (**TBFIELD (p. 1966)** command)
- User-defined friction (*TB0PT* = USER)

NPTS :

Number of data points to be specified for user-defined friction (*TB0PT* = USER). Not used for *TB0PT* = ISO or *TB0PT* = ORTHO.

TB0PT :

Friction options:

ISO --

Isotropic friction (one coefficient of friction, MU) (default). This option is valid for all 2D and 3D contact elements.

ORTHO --

Orthotropic friction (two coefficients of friction, MU1 and MU2). This option is valid for the following 3D contact elements: CONTA174, CONTA175, and CONTA177.

FORTHO --

Orthotropic friction (two coefficients of friction, MU1 and Mu2) with a friction coordinate system fixed in space. This option is valid for the following 3D contact elements: CONTA174, CONTA175, and CONTA177.

EORTHO --

Equivalent orthotropic friction (two coefficients of friction, MU1 and MU2). This option differs from *TB0PT* = ORTHO only in the way the friction coefficients are interpolated when they

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are dependent upon the following field variables: sliding distance and/or sliding velocity. In this case, the total magnitude of the field variable is used to do the interpolation.

USER --

User defined friction. This option is valid for all 2D and 3D contact elements.

References:

Contact Friction in the *Material Reference*

See also **TBFIELD (p. 1966)** for more information about defining a coefficient of friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity.

GASKET -- Gasket Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. The maximum number of temperatures specified is such that *NTEMP* * *NPTS* < 2000.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 5 for *TBOPT* = PARA. Default = 1 for all other values of *TBOPT*.

***TBOPT*:**

Gasket material options.

PARA --

Gasket material general parameters.

COMP --

Gasket material compression data.

LUNL --

Gasket linear unloading data.

NUNL --

Gasket nonlinear unloading data.

TSS --

Transverse shear data.

TSMS --

Transverse shear and membrane stiffness data. (If selected, this option takes precedence over TSS.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References:

Gasket in the *Material Reference*

Gasket Joints Simulation in the *Structural Analysis Guide*

GURSON -- Gurson Plasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. Maximum = 40.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

GURSON material options.

BASE --

Basic model without nucleation or coalescence (default).

SNNU --

Strain controlled nucleation.

SSNU --

Stress controlled nucleation.

COAL --

Coalescence.

References:

Gurson in the *Material Reference*

Gurson's Model in the *Theory Reference*

HFLM -- Film Coefficient Data Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. Maximum = 20.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 1. Maximum = 100.

***TB0PT*:**

Not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References:

FLUID116

HILL -- Hill Plasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. Maximum = 40.

***NPTS*:**

Not used.

***TB0PT* :**

Hill plasticity option:

(blank) --

Use one set of Hill parameters (default).

PC --

Enter separate Hill parameters for plasticity and creep. This option is valid for material combinations of creep and Chaboche nonlinear kinematic hardening only.

References:

Hill Anisotropy in the *Material Reference*

Hill Yield Criterion in the *Material Reference*

See also Combining Material Models in the *Material Reference*.

HYPER -- Hyperelasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. The maximum value of *NTEMP* is such that *NTEMP* x *NPTS* = 1000.

***NPTS*:**

Number of material parameters to be specified for a given temperature. Exceptions are for *TB0PT* = FOAM, OGDEN, POLY and YEOH, where *NPTS* is the number of terms in the material model's energy function.

***TB0PT*:**

Hyperelastic material options.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BOYCE--

Arruda-Boyce model. For *NPTS*, default = 3 and maximum = 3.

BLATZ --

Blatz-Ko model. For *NPTS*, default = 1 and maximum = 1.

ETUBE --

Extended tube model. Five constants (*NPTS* = 5) are required.

EXF1 --

Embedded fiber directions. Three constants (*NPTS* = 3) define the direction for *each* fiber. Up to five fibers (*NPTS* = 15) are allowed.

EX1 --

Embedded fiber strain energy potential. Two constants (*NPTS* = 2) are used for *each* fiber corresponding to the defined fiber directions. Undefined values default to zero.

EXA1 --

Embedded fiber compression strain energy potential. Two constants (*NPTS* = 2) are used for *each* fiber corresponding to the defined fiber directions. If not defined, the values specified via EX1 are used for both tension and compression.

FOAM --

Hyperfoam (Ogden) model. For *NPTS*, default = 1 and maximum is the number of terms in the energy function

GENT --

Gent model. For *NPTS*, default = 3 and maximum = 3.

HENCKY --

Hencky model. For *NPTS*, default = 2 and maximum = 2.

MOONEY --

Mooney-Rivlin model (default). You can choose a two-parameter Mooney-Rivlin model with *NPTS* = 2 (default), or a three-, five-, or nine-parameter model by setting *NPTS* equal to one of these values.

NEO --

Neo-Hookean model. For *NPTS*, default = 2 and maximum = 2.

OGDEN --

Ogden model. For *NPTS*, default = 1 and maximum is the number of terms in the energy function.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

POLY --

Polynomial form model. For *NPTS*, default = 1 and maximum is the number of terms in the energy function.

RESPONSE --

Experimental response function model. For *NPTS*, default = 0 and maximum is such that *NTEMP* x *NPTS* + 2 = 1000.

YEOH --

Yeoh model. For *NPTS*, default = 1 and maximum is the number of terms in the energy function.

USER --

User-defined hyperelastic model.

References:

Hyperelasticity in the *Material Reference*

INTER -- Contact Interaction Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1. No maximum limit. *NTEMP* is used only for user-defined contact interaction (*TBOPT* = USER).

***NPTS*:**

Number of data points to be specified. *NPTS* is used only for user-defined contact interaction (*TBOPT* = USER).

***TBOPT*:**

Contact interaction options.

The following options are valid only for general contact interactions specified via the GCDEF (p. 788) command:

STANDARD --

Standard unilateral contact (default).

ROUGH --

Rough, no sliding.

NOSEPE --

No separation (sliding permitted).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BONDED --

Bonded contact (no separation, no sliding).

ANOSEP--

No separation (always).

ABOND --

Bonded (always).

IBOND --

Bonded (initial contact).

The following option is valid for all 2D and 3D contact elements:

USER --

User-defined contact interaction.

References:

Contact Interaction in the *Material Reference*

Defining Your Own Contact Interaction (USERINTER) in the *Contact Technology Guide*

JOIN -- Joint Element Specifications

NTEMP:

Number of temperatures for which data will be provided. Default = 1.

NPTS:

Number of data points to be specified for a given temperature. *NPTS* is ignored if *TBOPT* = STIF or DAMP.

If Coulomb friction is specified, *NPTS* is used only for *TBOPT* = MUS1, MUS4, and MUS6.

TBOPT:

Joint element material options.

Linear stiffness behavior:

STIF --

Linear stiffness.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nonlinear stiffness behavior:

JNSA --

Nonlinear stiffness behavior in all available components of relative motion for the joint element.

JNS1 --

Nonlinear stiffness behavior in local UX direction only.

JNS2 --

Nonlinear stiffness behavior in local UY direction only.

JNS3 --

Nonlinear stiffness behavior in local UZ direction only.

JNS4 --

Nonlinear stiffness behavior in local ROTX direction only.

JNS5 --

Nonlinear stiffness behavior in local ROTY direction only.

JNS6 --

Nonlinear stiffness behavior in local ROTZ direction only.

Linear damping behavior:

DAMP --

Linear damping.

Nonlinear damping behavior:

JNDA --

Nonlinear damping behavior in all available components of relative motion for the joint element.

JND1 --

Nonlinear damping behavior in local UX direction only.

JND2 --

Nonlinear damping behavior in local UY direction only.

JND3 --

Nonlinear damping behavior in local UZ direction only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

JND4 --

Nonlinear damping behavior in local ROTX direction only.

JND5 --

Nonlinear damping behavior in local ROTY direction only.

JND6 --

Nonlinear damping behavior in local ROTZ direction only.

Friction Behavior:

Coulomb friction coefficient -

The values can be specified using either **TBDATA (p. 1960)** (*NPTS* = 0) or **TBPT (p. 1992)** (*NPTS* is nonzero).

MUS1 --

Coulomb friction coefficient (stiction) in local UX direction only.

MUS4 --

Coulomb friction coefficient (stiction) in local ROTX direction only.

MUS6 --

Coulomb friction coefficient (stiction) in local ROTZ direction only, *or*

Coulomb friction coefficient (stiction) for Spherical Joint.

Coulomb friction coefficient - Exponential Law -

Use **TBDATA (p. 1960)** to specify μ_s, μ_d, and c for the exponential law.

EXP1 --

Exponential law for friction in local UX direction only.

EXP4 --

Exponential law for friction in local ROTX direction only.

EXP6 --

Exponential law for friction in local ROTZ direction only.

Elastic slip:

SL1 --

Elastic slip in local UX direction only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SL4 --

Elastic slip in local ROTX direction only.

SL6 --

Elastic slip in local ROTZ direction only, *or*

Elastic slip for Spherical Joint.

TMX1 --

Critical force in local UX direction only.

TMX4 --

Critical moment in local ROTX direction only.

TMX6 --

Critical moment in local ROTZ direction only.

Stick-stiffness:

SK1 --

Stick-stiffness in local UX direction only.

SK4 --

Stick-stiffness in local ROTX direction only.

SK6 --

Stick-stiffness in local ROTZ direction only, *or*

Stick-stiffness for Spherical Joint.

Interference fit force/moment:

FI1 --

Interference fit force in local UX direction only.

FI4 --

Interference fit moment in local ROTX direction only.

FI6 --

Interference fit moment in local ROTZ direction only.

References:

MPC184 Joint in the *Material Reference*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

JROCK -- Jointed Rock Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

***TB0PT*:**

BASE --

Base material parameters.

RCUT --

Base material tension cutoff.

RSC --

Residual strength coupling.

FPLANE --

Joint parameters.

FTCUT --

Joint tension cutoff.

FORIE --

Joint orientation.

MSOL --

Material solution option.

References:

Jointed Rock in the *Material Reference*

MC -- Mohr-Coulomb Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***TB0PT*:**

BASE --

Mohr-Coulomb material parameters.

RCUT --

Tension cutoff.

RSC --

Residual strength coupling.

POTN --

Plastic potential.

FRICTION --

Friction angle scaling.

COHESION --

Cohesion scaling.

TENSION --

Tension strength scaling.

DILATATION --

Dilatancy angle scaling.

MSOL --

Material solution option.

References:

Mohr-Coulomb in the *Material Reference*

MELAS – Multilinear Elasticity Specifications

***NTEMP*:**

Number of temperatures for which data will be provided.

***NPTS*:**

Number of data points to be specified for a given temperature.

***TB0PT*:**

Not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References:

Multilinear Elasticity in the *Material Reference*.

MIGR – Migration Model Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

***TB0PT*:**

Migration model options.

0 --

Atomic (or ion) flux (default).

1 --

Vacancy flux.

References:

Migration Model in the *Material Reference*

Electric-Diffusion Analysis in the *Coupled-Field Analysis Guide*

Thermal-Diffusion Analysis in the *Coupled-Field Analysis Guide*

Structural-Diffusion Analysis in the *Coupled-Field Analysis Guide*

Electric-Diffusion Coupling in the *Theory Reference*

Thermal-Diffusion Coupling in the *Theory Reference*

Structural-Diffusion Coupling in the *Theory Reference*

MPLANE -- Microplane Specifications

***NTEMP*:**

The number of temperatures for which data will be provided. Default = 1. Maximum is such that *NTEMP* x *NPTS* = 1000.

***NPTS*:**

The number of data points to be specified for a given temperature. Default = 6. Maximum is such that *NTEMP* x *NPTS* = 1000.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***TB0PT*:**

Microplane model options:

ORTH --

Elastic microplane material with damage model (default).

DPC --

Coupled damage-plasticity microplane model.

NLOCAL --

Nonlocal parameters.

References:

Microplane in the *Material Reference*

NLISO -- Nonlinear Isotropic Hardening Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 4. Maximum = 4.

***TB0PT*:**

Isotropic hardening options.

VOCE --

Voce hardening law (default).

POWER --

Power hardening law.

References:

Nonlinear Isotropic Hardening in the *Material Reference*

PERF -- Perforated Material Specifications

NTEMP :

Not used.

NPTS :

Not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TB0PT :

Equivalent fluid model options:

JCA

Johnson-Champoux-Allard model

DLB

Delaney-Bazley model

MIKI

Miki model

ZPRO

Complex impedance and propagating constant model

CDV

Complex density and velocity model

Poroelastic acoustic material:

PORO

Poroelastic material model

Transfer admittance matrix options:

YMAT

General transfer admittance matrix model

SGYM

Transfer admittance matrix model of square grid structure

HGYM

Transfer admittance matrix model of hexagonal grid structure

References:

Perforated Media in the *Material Reference*

Equivalent Fluid of Perforated Materials in the *Theory Reference*

Poroelastic Acoustics in the *Theory Reference*

Perforated Material in the *Acoustic Analysis Guide*

Trim Element with Transfer Admittance Matrix in the *Acoustic Analysis Guide*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See **TBFIELD (p. 1966)** for more information about defining temperature and/or frequency-dependent properties.

PIEZ -- Piezoelectric Matrix Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

***TB0PT*:**

Piezoelectric matrix options.

0 --

Piezoelectric stress matrix [e] (used as supplied)

1 --

Piezoelectric strain matrix [d] (converted to [e] form before use)

References:

Piezoelectricity in the *Material Reference*

Piezoelectric Analysis

PLASTIC -- Nonlinear Plasticity Specifications

NTEMP :

Not used.

NPTS :

Not used.

TB0PT :

Plasticity option:

BISO --

Bilinear isotropic hardening plasticity.

BKIN –

Bilinear kinematic hardening plasticity.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MISO –

Multilinear isotropic hardening plasticity.

KINH --

Multilinear kinematic hardening plasticity.

The number of points (**TBPT (p. 1992)** commands issued) is limited to 100 for this option.

KSR2 --

Kinematic static recovery.

ISR --

Isotropic static recovery.

References:

Rate-Independent Plasticity in the *Material Reference*

PELAS -- Porous Elasticity Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

***TBOPT*:**

POISSON --

Porous elasticity model..

References:

Porous Elasticity in the *Material Reference*

PM -- Coupled Pore-Fluid Diffusion and Structural Model of Porous Media Specifications

***NTEMP*:**

The number of temperatures. Default = 1. The maximum must be a value such that (*NTEMP* x *NPTS*) <= 1000.

***NPTS*:**

The number of material constants. Default = 4. The maximum must be a value such that (*NTEMP* x *NPTS*) <= 1000.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***TB0PT*:**

Porous media options:

PERM --

Permeability

BIOT --

Biot coefficient

SP --

Solid property

FP --

Fluid property

DSAT --

Degree-of-saturation table

RPER --

Relative-permeability table

GRAV --

Gravity magnitude

References:

Porous Media Material Properties in the *Material Reference*

Porous Media Flow in the *Theory Reference*

Structural-Pore-Fluid-Diffusion-Thermal Analysis in the *Coupled-Field Analysis Guide*

Applying Initial Degree of Saturation and Relative Permeability in the *Advanced Analysis Guide*

See also VM260.

PRONY -- Prony Series Constant Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

Unused for *TB0PT* = EXPERIMENTAL.

***NPTS*:**

Defines the number of Prony series pairs for *TB0PT* = SHEAR or *TB0PT* = BULK. Default = 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unused for *TB0PT* = INTEGRATION and *TB0PT* = EXPERIMENTAL.

TB0PT :

Defines the behavior for viscoelasticity.

SHEAR--

Shear Prony series.

BULK --

Bulk Prony series.

INTEGRATION --

Stress update algorithm.

EXPERIMENTAL --

Complex modulus from experimental data.

References:

Viscoelasticity in the *Material Reference*

PZRS -- Piezoresistivity Specifications

NTEMP :

Not used.

NPTS :

Not used.

TB0PT :

Piezoresistive matrix options

0 --

Piezoresistive stress matrix (used as supplied)

1 --

Piezoresistive strain matrix (used as supplied)

References:

Piezoresistivity in the *Material Reference*

Piezoresistive Analysis in the *Coupled-Field Analysis Guide*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

RATE -- Rate-Dependent Plasticity Specifications

***NTEMP*:**

The number of temperatures for which data will be provided. Default is 1. Maximum is such that *NTEMP* x *NPTS* = 1000.

***NPTS*:**

The number of data points to be specified for a given temperature. Default = 2. Maximum is such that *NTEMP* x *NPTS* = 1000.

***TB0PT*:**

Rate-dependent viscoplasticity options.

PERZYNA --

Perzyna option (default).

PEIRCE --

Peirce option.

EVH --

Exponential visco-hardening option.

ANAND --

Anand option.

References:

Rate-Dependent Plasticity (Viscoplasticity) in the *Material Reference*

Viscoplasticity Model in the *Structural Analysis Guide*

Rate-Dependent Plasticity in the *Theory Reference*

See also Combining Material Models in the *Material Reference*.

RO -- Ramberg-Osgood Specifications

NTEMP :

Number of temperatures for which data will be provided. Default = 1.

NPTS :

Number of properties to be defined for the material option: 3 (required).

TB0PT :

Not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References:

Ramberg-Osgood Model in the *Material Reference*

SDAMP -- Material Damping Coefficient Specifications

NTEMP :

Number of temperatures for which data will be provided. Default = 1.

NPTS :

Number of properties to be defined for the material option. Default = 1 for each material damping option (*TB0PT*) selected.

TB0PT :

Material damping options:

STRU or 1 --

Structural damping coefficient (default).

ALPD or 2--

Rayleigh mass proportional material damping.

BETD or 3--

Rayleigh stiffness proportional material damping.

References:

Material Damping in the *Material Reference*

Full Harmonic Analysis in the *Structural Analysis Guide*

Damping Matrices in the *Theory Reference*

SHIFT -- Shift Function Specifications

NTEMP :

Allows one temperature for which data will be provided.

NPTS :

Number of material constants to be entered as determined by the shift function specified via *TB0PT*. Not used for *TB0PT* = PLIN.

3 --

for *TB0PT* = WLF

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 --

for *TB0PT* = TN

n $_f$ **--**

for *TB0PT* = FICT, where n_f is the number of partial fictive temperatures

TB0PT :

Shift function:

WLF --

Williams-Landel-Ferry.

TN --

Tool-Narayanaswamy.

FICT --

Tool-Narayanaswamy with fictive temperature.

PLIN --

Piecewise linear.

USER --

User-defined.

References:

Viscoelasticity in the *Material Reference*

SINT -- Sintering Specifications

NTEMP :

Not used.

NPTS :

Not used.

TB0PT :

Sintering options:

INIT --

Initial conditions: relative density, particle diameter, and grain-size diameter. The initial relative density can alternatively be specified as a location-varying initial state (**INISTATE (p. 938)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PARAM --

Sintering activation temperature and mode specification.

STRESS --

Sintering stress coefficients.

VSCOEF --

Viscosity coefficients. Mutually exclusive with VSTABLE.

VSTABLE --

Table of viscosity values. Mutually exclusive with VSCOEF.

GROWTH --

Grain-growth parameters.

RIEDEL --

Selects the Riedel sintering model (default) and defines the viscous moduli coefficients.

SOVS --

Selects the Skorohold-Olevsky sintering model and defines the viscous moduli coefficients.

ANICONST --

Orthotropic factors to be applied to the viscous bulk and shear moduli. The factors remain constant throughout densification.

References:

Sintering in the *Material Reference*

SMA -- Shape Memory Alloy Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 7 if *TB0PT* = SUPE or MEFF, 2 if *TB0PT* = METE, 6 if *TB0PT* = METL or METH, and 7 if *TB0PT* = MEPD.

***TB0PT*:**

Shape memory model option:

SUPE -- Superelasticity option (default).

MEFF -- Shape memory effect option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

METE – Shape memory effect with plasticity option: elastic phase-dependent and thermal expansion.

METL – Shape memory effect with plasticity option: limits of transformation in strain-stress-temperature space.

METH – Shape memory effect with plasticity option: transformation hardening.

MEPD – Shape memory effect with plasticity option: plastic response.

METC – Shape memory effect with plasticity option: tension-compression asymmetry response and hysteresis response.

Reference:

Shape Memory Alloy (SMA) in the *Material Reference*

SOIL -- Soil Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

***TB0PT*:**

CAMCLAY --

Modified Cam-clay material model.

MSOL --

Material solution option.

References:

Cam-clay in the *Material Reference*

STATE -- User-Defined State Variable Specifications

When *Lab* = STATE, state variable specifications affect user-defined material models. The subroutine in use depends on the element type used when *Lab* = USER is specified.

***NTEMP*:**

Not used.

***NPTS*:**

Number of state variables.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***TB0PT*:**

Not used.

References:

Customizing Material Behavior in the *Material Reference*

SWELL -- Swelling Specifications

***NTEMP*:**

Number of temperatures for which data will be provided. The maximum value of *NTEMP* is such that *NTEMP* x *NPTS* = 1000

***NPTS*:**

Number of data points to be specified for a given temperature. The maximum value of *NPTS* is such that *NPTS* x *NTEMP* = 1000.

***TB0PT*:**

Swelling model options:

LINEAR --

Linear swelling function.

EXPT --

Exponential swelling function.

USER --

User-defined swelling function. Define the swelling function via subroutine `userswstrain` (described in the *Programmer's Reference*). Define temperature-dependent constants via the **TBTEMP (p. 1993)** and **TBDATA (p. 1960)** commands. For solution-dependent variables, define the number of variables via the **TB (p. 1899)**,STATE command.

References:

Swelling in the *Material Reference*

Swelling Model in the *Structural Analysis Guide*

THERM -- Thermal Properties Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***TB0PT*:**

Thermal properties:

COND --

Thermal conductivity.

ENTH --

Enthalpy. Enthalpy must be a function of temperature only (see Considerations for Enthalpy (p. 1955)).

SPHT --

Specific heat. For porous media, solid-skeleton specific heat.

FLSPHT --

Fluid-specific heat for porous media.

References:

Thermal Properties in the *Material Reference*

Porous Media Mechanics in the *Material Reference*

TNM -- Three-Network Model Properties Specifications

***NTEMP*:**

Not used.

***NPTS*:**

Not used.

***TB0PT*:**

Three-network model material options:

NETA --

Network A properties.

NETB --

Network B properties.

NETC --

Network C properties.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FLOW --

Network flow properties.

TDEP --

Temperature-dependence factors.

LOCK --

Chain-locking stretch.

BULK --

Bulk modulus.

References:

Three-Network Model (**TB**,TNM) in the *Material Reference*

USER -- User-Defined Material Model or Thermal Material Model Specifications

When *Lab* = USER, the **TB (p. 1899)** command activates either the `UserMat` (user-defined material) or the `UserMatTh` (user-defined thermal material) subroutine automatically. The subroutine activated depends on the element type used and the *TB0PT* setting. For a detailed list of elements that support **TB (p. 1899)**,USER, see Material Model Support for Elements in the *Material Reference*..

***NTEMP*:**

Number of temperatures for which data will be provided. Default = 1.

***NPTS*:**

Number of data points to be specified for a given temperature. Default = 48.

***TB0PT*:**

User-defined material model (`UserMat`) or thermal material model (`UserMatTh`) options:

NONLINEAR

Nonlinear iterations are applied (default except for thermal elements).

LINEAR

Nonlinear iterations are not applied. This option is ignored if there is any other nonlinearity involved, such as contact, geometric nonlinearity, etc.

MXUP

This option indicates a UserMat material model to be used with mixed u-P element formulation for material exhibiting incompressible or nearly incompressible behavior.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

THERM

Invokes the thermal material model (UserMatTh) for a coupled-field analysis using elements SOLID225, SOLID226, and SOLID227 with thermal degrees of freedom. Use this option in a coupled structural-thermal analysis to specify a user-defined thermal material model (UserMatTh) independently of the user-defined structural material model (UserMat).

Invokes the thermal material model (UserMatTh) for thermal elements SOLID278, SOLID279, SOLID291, PLANE292, and PLANE293. For thermal elements, this is the only option (no default).

References:

Customizing Material Behavior in the *Material Reference*

Subroutine UserMat (Creating Your Own Material Model) in the *Programmer's Reference*

Subroutine UserMatTh (Creating Your Own Thermal Material Model) in the *Programmer's Reference*

WEAR -- Contact Surface Wear Specifications

NTEMP :

Number of temperatures for which data will be provided.

NPTS :

Number of data points to be specified for the wear option. This value is set automatically based on the selected wear option (*TB0PT*). If *TB0PT* is not specified, the default becomes *NPTS* = 5 and *TB0PT* = ARCD.

TB0PT :

Wear model options:

ARCD --

Archard wear model (default).

USER --

User-defined wear model.

AUTS --

Automatic scaling of wear increment. Must be used in conjunction with one of the wear models (*TB0PT* = ARCD or USER).

CBCS --

Autoscaling of wear increment over each simulation cycle. Must be used along with one of the wear models (*TB0PT* = ARCD or USER)..

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

References:

Contact Surface Wear in the *Material Reference*

Contact Surface Wear in the *Contact Technology Guide*

See also **TBFIELD (p. 1966)** for more information about defining temperature and/or time-dependent properties.

XTAL -- Crystal Plasticity Model Specifications

NTEMP :

Unused.

NPTS :

Unused.

TB0PT :

Crystal plasticity material options:

ORIE --

Crystal orientation.

NSLFAM --

Number of slip families.

FORM --

Formulation number.

XPARAM --

Crystal characteristic parameters.

HARD --

Slip system hardness properties.

FLFCC --

Face-centered cubic (FCC) flow parameters.

FLHCP --

Hexagonal closed packed (HCP) flow parameters.

FLBCC --

Body-centered cubic (BCC) flow parameters.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference:

Crystal Plasticity in the *Material Reference*

Notes

TB (p. 1899) activates a data table for use by subsequent **TBDATA (p. 1960)** or **TBPT (p. 1992)** commands. The table space is initialized to zero values. Data from this table are used for most nonlinear material descriptions as well as for special input for some elements.

For a list of elements supporting each material model (*Lab* value), see Material Model Support for Elements in the *Material Reference*.

For information about linear material property input, see **MP (p. 1160)**.

This command is also valid in SOLUTION.

Considerations for Enthalpy (*TBOPT* = ENTH)

- To ensure correct results, you must define enthalpy over a large enough temperature range to span all computed temperatures during the solution. The **TB (p. 1899)** command does not extrapolate enthalpy values beyond the specified temp range like the **MP (p. 1160)** command does.
- If both the **TB (p. 1899)** and **MP (p. 1160)** commands are used to specify enthalpy values, enthalpy values defined via the **TB (p. 1899)** command are used and those defined via the **MP (p. 1160)** command are ignored.

Product Restrictions

Material Model (*Lab* or *TBOPT*)[a]	Description	Available Products
AFDM	Acoustic frequency-dependent material (p. 1905)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
AHYPER	Anisotropic hyperelasticity (p. 1906)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ANEL	Anisotropic elasticity (p. 1907)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
AVIS	Anisotropic viscosity (p. 1908)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BB	Bergstrom-Boyce (p. 1908)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BH	Magnetic field (p. 1909)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BISO (specified as **TB (p. 1899)**,PLASTIC (p. 1941),,,,BISO)	Bilinear isotropic hardening (p. 1941)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
BKIN (specified as **TB (p. 1899)**,PLASTIC (p. 1941),,,,BKIN)	Bilinear kinematic hardening (p. 1941)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CAST	Cast iron (p. 1910)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CDM	Damage (p. 1910)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CFOAM	Crushable foam (p. 1912)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CGCR	Crack-growth (p. 1912)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CHABOCHE	Chaboche nonlinear kinematic hardening (p. 1913)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CONCR	Concrete (p. 1914)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CREEP	Creep (p. 1915)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CTE	Coefficient of thermal expansion (p. 1916)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
CZM	Cohesive zone (p. 1917)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DLST	Anisotropic dielectric loss tangent (p. 1919)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DMGE	Damage evolution law (p. 1919)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DMGI	Damage initiation criteria (p. 1920)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
DPER	Anisotropic electric permittivity (p. 1920)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
EDP	Extended Drucker-Prager (p. 1921)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ELASTIC	Elasticity (p. 1922)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ELASTIC (*TB0PT* = USER)	Elasticity (p. 1922) (user-defined)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ELST	Anisotropic elastic loss tangent (p. 1923)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
EXPE	Experimental data (p. 1923)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
FCON	Fluid conductance data (p. 1924)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
FCLI	Material strength limits (p. 1925)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
FLUID	Fluid (p. 1925)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FRIC	Coefficient of friction (p. 1926)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
GASKET	Gasket (p. 1927)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
GURSON	Gurson pressure-dependent plasticity (p. 1928)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
HFLM	Film coefficient data (p. 1928)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
HILL	Hill anisotropy (p. 1929)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
HYPER	Hyperelasticity (p. 1929)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
HYPER (*TB0PT* = USER)	Hyperelasticity (p. 1929) (user-defined)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
INTER	Contact interaction (p. 1931)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
INTER (*TB0PT* = USER)	Contact interaction (p. 1931) (user-defined)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
ISR (specified as PLASTIC (p. 1941),,,,ISR)	Isotropic static recovery (p. 1941)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
JOIN	Joint (p. 1932)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
JOIN (*TB0PT* = STIF)	Joint (p. 1932) (user-defined)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
JROCK	Jointed rock (p. 1936)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
KINH (specified as PLASTIC (p. 1941),,,,KINH)	Multilinear kinematic hardening plasticity (p. 1941)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
KSR2 (specified as PLASTIC (p. 1941),,,,KSR2)	Kinematic static recovery (p. 1941)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MC	Mohr-Coulomb (p. 1936)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MELAS	Multilinear elasticity (p. 1937)	Pro | **Premium** | **Enterprise** | PrepPost | Solver | AS add-on
MIGR	Migration (p. 1938)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MISO (specified as PLASTIC (p. 1941),,,,MISO)	Multilinear isotropic hardening plasticity (p. 1941)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MPLANE	Microplane (p. 1938)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
NLISO	Voce isotropic hardening law (p. 1939) (power law)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PELAS	Porous elasticity (p. 1942)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PERF	Perforated material (p. 1939)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PIEZ	Piezoelectric matrix (p. 1941)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PLASTIC	Nonlinear plasticity (p. 1941)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PM	Porous media (p. 1942)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PRONY	Prony series (p. 1943)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
PZRS	Piezoresistivity (p. 1944)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
RATE	Rate-dependent plasticity (p. 1945)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
RO	Ramberg-Osgood (p. 1945)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SDAMP	Material damping coefficients (p. 1946)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SHIFT	Shift function (p. 1946)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SINT	Sintering (p. 1947)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SMA	Shape memory alloy (p. 1948)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SOIL	Soil models (p. 1949)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
STATE	User-defined state variables (p. 1949)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
SWELL	Swelling (p. 1950)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
THERM	Thermal (p. 1950)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
TNM	Three-network model (p. 1951)	Pro | Premium | **Enterprise** | PrepPost | Solver | AS add-on
USER	User-defined (p. 1952)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
WEAR	Contact surface wear (p. 1953)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on
XTAL	Crystal Plasticity (p. 1954)	Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

[a] **TB (p. 1899)**,*Lab* value or **TB (p. 1899)**,*Lab*,,,,*TB0PT* value.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBCOPY, *Lab*, *MATF*, *MATT*

Copies a data table from one material to another.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Data table label. See the **TB (p. 1899)** command for valid labels, and see "Notes" (p. 1959) for *Lab* = ALL.

MATF

Material reference number where data table is to be copied from.

MATT

Material reference number where data table is to be copied to.

Notes

The **TBCOPY (p. 1959)** command, with *Lab* = ALL, copies all of the nonlinear data defined by the **TB (p. 1899)** command. If you copy a model that includes both yield behavior constants and linear constants (for example, a BKIN model), **TBCOPY (p. 1959)**,ALL and **MPCOPY (p. 1167)** are used together to copy the entire model. All input data associated with the model is copied, that is, all data defined through the **TB (p. 1899)** and **MP (p. 1160)** commands.

Also, if you copy a material model using the Material Model Interface (**Edit> Copy**), both the commands **TBCOPY (p. 1959)**,ALL and **MPCOPY (p. 1167)** are issued, regardless of whether the model includes linear constants only, or if it includes a combination of linear and yield behavior constants.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TBDATA, *STLOC*, *C1*, *C2*, *C3*, *C4*, *C5*, *C6*

Defines data for the material data table.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

STLOC

Starting location in table for entering data. For example, if *STLOC* = 1, data input in the *C1* field applies to the first table constant, *C2* applies to the second table constant, etc. If *STLOC*=5, data input in the *C1* field applies to the fifth table constant, etc. Defaults to the last location filled + 1. The last location is reset to 1 with each **TB (p. 1899)**, **TBTEMP (p. 1993)**, or **TBFIELD (p. 1966)** command.

C1, C2, C3, . . . , C6

Data values assigned to six locations starting with *STLOC*. If a value is already in this location, it is redefined. A blank value leaves the existing value unchanged.

Notes

Defines data for the table specified via the most recent **TB (p. 1899)** command (at the temperature specified via the most recent **TBTEMP (p. 1993)** or **TBFIELD (p. 1966)** command, if applicable).

The type of data table specified determines the number of data values needed in **TBDATA (p. 1960)**. Data values are interpolated for temperatures or other specified field variables that fall between user-defined **TBTEMP (p. 1993)** or **TBFIELD (p. 1966)** values.

You can specify values for up to six constants per **TBDATA (p. 1960)** command. Issue the command multiple times if needed.

Some elements (for example, SOLID226) support tabular input for some linear materials. For a list of elements supporting tabular material properties and associated primary variables, see Defining Linear Material Properties Using Tabular Input in the *Material Reference*.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TBDELE, *Lab*, *MAT1*, *MAT2*, *INC*, *TBOPT*

Deletes previously defined material data tables.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Material data table label to delete. (See **TB (p. 1899)** for valid *Lab* values.)

MAT1*, *MAT2*, *INC

Deletes data tables for materials *MAT1* to *MAT2* (default = *MAT1*) in steps of *INC* (default = 1).

TBOPT

Material data table option. (See **TB (p. 1899)** for valid *TBOPT* values for the given *Lab*.)

Notes

If *Lab* = ALL, delete *all* material data tables.

If *MAT1*= ALL, *MAT2* and *INC* are ignored and all material data tables are deleted.

If *TBOPT* is specified, the material data table corresponding to *Lab* is deleted if it also has the specified table option. If *TBOPT* is not specified, all material data tables corresponding to *Lab* are deleted. *TBOPT* is ignored when *Lab* = ALL.

This command is also valid in SOLUTION, but is not intended for changing material behaviors between load steps.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

TBEO, *Par*, *Value*

Sets special options or parameters for material data tables.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Par

Parameter name:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CAPCREEPREG

Available for the viscoplasticity/creep model (**TB (p. 1899)**,CREEP (p. 1915)), allows two creep models to be specified via the same material ID when used with the Extended Drucker-Prager model (**TB (p. 1899)**,EDP (p. 1921)).

FDCS

Coordinate system to use with location (XCOR, YCOR, ZCOR) or displacement (UX, UY, UZ) field variables.

NEGSLOPE

Controls whether negative tangent slopes of the stress-strain curve are allowed for multi-linear kinematic or isotropic hardening in a rate-independent plasticity analysis.

Value

Parameter value:

When *Par* = CAPCREEPREG --

SHEA

Use the shear stress-state creep model with the Extended Drucker-Prager model.

COMP

Use the compaction stress-state creep model with the Extended Drucker-Prager model.

When *Par* = FDCS --

Any predefined, user-defined, or custom (**LOCAL (p. 1059)** or **CS (p. 435)**) Cartesian coordinate system number.

When *Par* = NEGSLOPE --

0

Error-trap negative tangent slopes of the stress-strain curve (default).

1

Allow negative tangent slopes of the stress-strain curve.

Notes

Issue the **TBEO (p. 1961)** command *after* activating the material data table (**TB (p. 1899)**) but *before* defining data for the table (**TBDATA (p. 1960)**) or a point on a nonlinear data curve (**TBPT (p. 1992)**).

If the defined material data table has subtables, issue the **TBEO (p. 1961)** command for *each* desired subtable.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

TBIN,*Oper*,*Par1*,*Par2*,*Par3*,*Par4*

Sets parameters used for interpolation of the material data tables.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Operation to perform:

ALGO (p. 1963)

Specifies the interpolation algorithm to use for the subtable (or table if the material data table has only one subtable) being defined.

BNDS (p. 1964)

Specifies the maximum and minimum bounds for individual field variables.

CACH (p. 1964)

Enables or disables caching of interpolated data for better performance.

DEFA (p. 1964)

Specifies the default value of the user-defined field variable used for interpolation (if no value was specified).

EXTR (p. 1964)

Controls extrapolation options.

NORM (p. 1965)

Scales the field variables before interpolation.

SCALE (p. 1965)

Interpolates **TB (p. 1899)**-based material parameters in the linear- or natural-log scale.

Interpolation Parameters for *Oper* = ALGO

Par1

Interpolation algorithm:

LINEAR – Linear 1D / 2D (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LMUL – Linear-multivariate interpolation (multidimensional).

Par1 = LINEAR is available for all material models. The remaining options are limited to a subset of material models. For more information, see Material Model Support for Interpolation in the *Material Reference*.

Interpolation Parameters for *Oper* = BNDS

Par1

The field variable on which the operation is being applied.

Par2

Lower bound of the field variable.

Par3

Upper bound of the field variable.

Interpolation Parameters for *Oper* = CACH

Par1

Reserved for future use.

Par2

Enable or disable caching of interpolated material parameters. Enable for better performance.

OFF – Disable (default).

ON – Enable.

Interpolation Parameters for *Oper* = DEFA

Par1

The field variable on which the operation is being applied.

Par2

Default value of the field variable for which an initial value was not specified.

Interpolation Parameters for *Oper* = EXTR

Par1

Reserved for future use.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Par2

Set extrapolation/projection options for interpolating material parameters.

OFF / BBOX– Projects to the hyper-rectangular bounding box (default). An error occurs if query points exist outside the convex hull of points but inside the hyper-rectangular bounding box.

PHULL – Projects to the convex hull of points if a point is located outside the convex hull surface.

TBIN (p. 1963),EXTR is supported for the linear multivariate interpolation algorithm only.

Interpolation Parameters for *Oper* = NORM

Par1

Reserved for future use.

Par2

Enable or disables field-variable normalization for interpolation.

OFF – Disable.

ON – Enable (default).

Interpolation Parameters for *Oper* = SCALE

Par1

Independent variable, which can be any field variable specified via the **TBFIELD (p. 1966)** command.

Par2

Index of any material parameter specified via the **TBDATA (p. 1960)** command.

Par3

Scale to use for the independent variable. Valid options are LINEAR (linear) or LOG (logarithmic).

Par4

Scale to use for the dependent variable (the material parameter specified via *Par2*). Valid options are LINEAR (linear) or LOG (logarithmic).

Notes

For a list of the supported material data tables (**TB (p. 1899)**), see Logarithmic Interpolation and Scaling in the *Material Reference*.

Oper = DEFA, BNDS, NORM and CACH are supported for the linear multivariate (**TBIN (p. 1963)**,ALGO,LMUL) interpolation algorithm only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

TBFIELD, *Type*, *Value*

Defines values of field variables for material data tables.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Field variable type:

CYCLE

A healing cycle number is to be specified in *Value*.

FREQ

A frequency is to be specified in *Value*.

NPRES

A normal pressure is to be specified in *Value*.

PLSR

An equivalent plastic strain rate is to be specified in *Value*.

PPRE

Pressure degree of freedom is to be specified in *Value*.

SLDA

A total sliding distance (algebraic) is to be specified in *Value*.

SLDI

A total sliding distance (absolute) is to be specified in *Value*.

SLRV

A sliding velocity is to be specified in *Value*.

SRAT

Stress ratio of fatigue load cycle is to be specified in *Value*.

TEMP

A temperature is to be specified in *Value*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TIME

A time is to be specified in *Value*.

UFXX

User-defined field variable (UF01,UF02, ..., UF09).

UX / UY / UZ

Displacements in the global/local X, Y, or Z coordinate system, respectively, are to be specified in *Value*.

XCOR / YCOR / ZCOR

X, Y and Z locations, respectively, are to be specified in *Value*..

Value

The field value to be referenced.

Notes

Define your data tables as field-variable-dependent (via the appropriate **TB (p. 1899)** command), then issue **TBFIELD (p. 1966)** to define the field values.

Issue this command multiple times to enter values for different field variables.

Define data values in ascending order for all field quantities. If a field value is to be held constant, define it only once; subsequent definitions are ignored.

No limit exists on the number of values that you can specify. The specified field value remains active until the next **TBFIELD (p. 1966)** command is input.

Field variables associated with **TB (p. 1899)** commands are supported only for current-technology structural and thermal elements.

After you have defined the field value(s), define your data for the data tables (**TBDATA (p. 1960)**).

For more information about the interpolation scheme used for field-dependent material properties, see Understanding Field Variables in the *Material Reference*.

For more information about using **TBFIELD (p. 1966)** with **TB (p. 1899)**,ELASTIC or **TB (p. 1899)**,SDAMP, see Full Harmonic Analysis in the *Structural Analysis Guide*.

Material Model Support for Field Variables

The TEMP (temperature) predefined field variable is available for all material models defined via **TB (p. 1899)**,*Lab*.

Several other field variables are available for use with some material models (when used with specific element types), such as TIME (time), PPRE (pore-pressure), XCOR / YCOR / ZCOR (location), UX / UY / UZ (displacement), and UF01 - UF09 (user-defined).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The field variables can be defined in the global coordinate system or in any local or user-defined coordinate system.

For more information, see Predefined Field Variables in the *Material Reference*.

Menu Paths

This command cannot be accessed from a menu.

TBFPLOT,*MATID*,*CurveFitName*,*EXPDATID*,*ColX*,*ColY1*,*ColY2*

Plots material curve-fitting data.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MATID

Material reference identification number.

'CurveFitName'

Material curve-fitting model name (obtainable via **TBFT (p. 1969)**,LIST). Enclose the name within single quotes.

EXPDATID

Experimental data ID.

ColX

Experimental data column to use in the X axis.

ColY1

Experimental data column to use in the Y axis.

ColY2

Fitted-data column to use in the Y axis.

Notes

This command plots the fitted data specified via *ColY2* (the number of experimental data columns + 1) and the data specified via *ColY1* as a function of the X-axis data specified via *ColX*.

Issue this command *after* curve-fitting has been completed (**TBFT (p. 1969)**,SOLVE).

Material curve-fitting does not support saving to (**SAVE (p. 1645)**) and resuming from (**RESUME (p. 1601)**) the database file. You must therefore rerun the curve-fitting analysis, then issue **TBFPLOT (p. 1968)** again to replot.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Material Props>Material Models

TBFT,*Oper*,*MATID*,*Option1*,*Option2*,*Option3*,*Option4*,*Option5*,*Option6*,*Option7*,–,–,*Option10*

Performs material curve-fitting operations.

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Use material curve-fitting to evaluate your experimental data and correlate it to the program's built-in nonlinear material models. You apply coefficients to your curve data, determine a fit with existing model, and write the data according to the table configurations described in the **TB (p. 1899)** command. Curve-fitting is available for nonlinear models associated with:

- Hyperelastic material behavior (p. 1969)
- Viscoelastic material behavior (p. 1972)
- Chaboche plasticity material behavior (p. 1976)
- Creep material behavior (p. 1980)
- Thermomechanical fatigue (TMF) and plasticity combinations (p. 1984)

TBFT Specifications for **Hyperelastic Models**

TBFT (p. 1969), *Oper*, *MATID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

Oper

The operation to perform:

Operation Set 1 (Curve-Fitting)

- - - - - - - - - - - - - - - - - - -

FADD --

Define a constitutive model for parameter-fitting and import all parameters (defined previously via **TB (p. 1899)** and **TBDATA (p. 1960)**).

FDEL --

Delete a constitutive model.

FSET --

Write data related to a constitutive model to the database (same as **TB (p. 1899)** command).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SET --

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

CDEL --

Deletes coefficients at current reference temperature. Applicable only for temperature dependent coefficients.

SOLVE --

Solve for coefficients.

FIX --

Fix (hold constant) the coefficient you specify in *Option4*.

Operation Set 2 (Experimental Data)

- - - - - - - - - - - - - - - - - - - -

EADD --

Add experimental data.

EDEL --

Delete experimental data.

Other Operations

- - - - - - - - - - - - - - - - - - - -

LIST --

List all data associated with the material model represented by the material ID number (*MATID*)

PSCA

Activate parameter-scaling to improve optimization performance. Valid for the UserMat curve-fitting method only (**TB (p. 1899)**,USER,,,,MXUP and TB,USER,,,,NONLINEAR [default]).

MATID

Material reference identification number (same as *MAT* argument used in the **TB (p. 1899)** command). Valid value is any number *n*, where 0 < *n* < 100,000. Default = 1.

Option1

For curve-fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE or FIX) this value specifies the category (AML).

For adding or deleting your experiment (*Oper* = EADD *or* EDEL), this value specifies the experimental data type. **Valid options:** UNIA (default), BIAX, SHEA, SSHE,and VOLU.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option2

For curve-fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE, or FIX), set this value to GENR.

When you need to specify a file name from which to get experimental data (*Oper* = EADD), place that string here. Valid value is any file name string. You can enter the entire `path\filename.extension` string and leave the next two values (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next value, and the path following.

Option3

For *Oper* = FADD, FDEL, FSET, CDEL, SET, SOLVE or FIX, set this value to a user-defined name (to be used consistently in the curve-fitting process).

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value will contain the file extension.

Option4

When you are working on a specific coefficient (*Oper* = FIX), this value specifies the index of that coefficient. Valid values vary from 1 to *n*, where *n* is the total number of coefficients. Default = 1.

For *Oper* = SET, see Table 227: Set Operations (p. 1971), below.

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value will contain the directory/path specification.

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid values are 0 for non-normalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

Option5

When you are working on a specific coefficient (*Oper* = FIX), this value specifies the index of that coefficient. Valid values vary from 1 to N, where N is the total number of coefficients. Default = 1.

For *Oper* = SET, see Table 227: Set Operations (p. 1971), below.

Table 227: Set Operations

Purpose	Option4	Option5
Set the value of the coefficient.	Index of coefficient	Value of that coefficient
Set temperature dependency ON/OFF[a]	TDEP	1 -- ON 0 -- OFF
Set reference temperature	TREF	Temperature value

[a] Specify temperature data in the same specified via **TREF (p. 2013)**.

If *Oper* = PSCA, use this value to specify the scaling option to be used in the parameter-fitting process. Options are LINE (default), EXPO (exponential), SQRD (squared) or LOG (logarthmic). Use with the UserMat curve-fitting method.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *Oper* = SOLVE, use this value to specify the number of iterations to be used in the calculation of the coefficients. Valid value is any positive integer. Default = 1000.

Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid value is 0.0 to 1.0. Default = 0.0.

Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid value is 0 to 1. Default = 0.

For the supported list of material models, see Valid Hyperelastic Curve-Fitting Model Types in the *Material Reference*.

This table summarizes the format for hyperelastic curve-fitting operations via **TBFT (p. 1969)**:

Hyperelastic Model Command Summary

Oper	MATID	Option1	Option2	Option3	Option4	Option5	Option6	Option7
FADD	*MATID*	AML	GENR	*UserDefName*	-	-	-	-
FDEL	*MATID*	AML	GENR	*UserDefName*	-	-	-	-
FSET	*MATID*	AML	GENR	*UserDefName*	-	-	-	-
SET	*MATID*	AML	GENR	*UserDefName*	*Index*	*Value*	-	-
SOLVE	*MATID*	AML	GENR	*UserDefName*	*NormFlag*	*NumIter*	RTOL	CTOL
FIX	*MATID*	AML	GENR	*UserDefName*	*Index*	-	-	-
EADD	*MATID*	*ExpTyp*	*File*	*UserDefName*	*Directory*	-	-	-
EDEL	*MATID*	*ExpType / Index*	-	*UserDefName*	-	-	-	-

TBFT Specifications for Viscoelastic Models

TBFT (p. 1969), *Oper*, *MATID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

Oper

The operation to perform:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operation Set 1 (Curve-Fitting)

- - - - - - - - - - - - - - - - - - -

FADD --

Define a constitutive model.

FDEL --

Delete a constitutive model.

FSET --

Write data related to a constitutive model to the database (same as **TB (p. 1899)** command).

SET --

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

CDEL --

Deletes coefficients at current reference temperature. Applicable only for temperature dependent coefficients.

SOLVE --

Solve for coefficients.

FIX --

Fix (hold constant) the coefficient you specify in *Option4*.

Operation Set 2 (Experimental Data)

- - - - - - - - - - - - - - - - - - -

EADD --

Add experimental data.

EDEL --

Delete experimental data.

Other Operations

- - - - - - - - - - - - - - - - - - -

LIST --

List all data associated with the material model represented by the material ID number (*MATID*)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MATID

Material reference identification number (same as *MAT* argument used in the **TB (p. 1899)** command). Valid value is any number *n*, where 0 < *n* < 100,000. Default = 1.

Option1

For curve-fitting function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE, or FIX), this value is set to AML.

For adding or deleting your experiment (*Oper* = EADD *or* EDEL), this value specifies the experiment type. **Valid options:** SDEC (Shear Modulus vs. Time/Freq) or BDEC (Bulk Modulus vs. Time/Freq).

Option2

For curve-fitting function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE, or FIX), this value is set to GENR.

When you need to specify a file name from which to get experimental data (*Oper* = EADD), place that string here. Valid value is any file name string. You can enter the entire `path\filename.extension` string and leave the next two values (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next value, and the path following.

Option3

For *Oper* = FADD, FDEL, FSET, CDEL, SET, SOLVE, or FIX, set this value to any user-defined name (to be used consistently during the curve-fitting process).

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value will contain the file extension.

Option4

When you are working on a specific coefficient (*Oper* = FIX), this value specifies the index of that coefficient. Valid values vary from 1 to *n*, where *n* is the total number of coefficients. Default = 1.

For *Oper* = SET, see Table 228: Set Operations (p. 1974) below.

You can also specify *TREF* to indicate the reference temperature.

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value will contain the directory/path specification.

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid values are 0 for non-normalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

Option5

For *Oper* = SET, see Table 228: Set Operations (p. 1974) below.

Table 228: Set Operations

Purpose	Option4	Option5
Set the value of the coefficient	Index of coefficient	Value of coefficient

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Purpose	Option4	Option5
Set temperature dependency ON/OFF	TDEP	1 for ON and 0 for OFF
Set reference temperature	TREF	Temperature value

If *Oper* = SOLVE, use this value to specify the number of iterations to be used in the calculation of the coefficients. Valid value is any positive integer. Default = 1000.

If you are specifying a coefficient to be held constant (*Oper* = FIX):

1 – Fix the specified coefficient.

0 – Allow the specified coefficient to vary (disable fixing).

Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid value is 0.0 to 1.0. Default = 0.0.

Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid value is 0 to 1. Default = 0.

This table summarizes the format for viscoelastic curve-fitting operations via **TBFT (p. 1969)**:

Viscoelastic Model Command Summary

Oper	**MAT-ID**	**Option1**	**Option2**	**Option3**	**Option4**	**Option5**	**Option6**	**Option7**
FADD	*MAT-ID*	AML	GENR	*UserDef-Name*	-	-	-	-
FDEL	*MAT-ID*	AML	GENR	*UserDef-Name*	-	-	-	-
FSET	*MAT-ID*	AML	GENR	*UserDef-Name*	-	-	-	-
SET	*MAT-ID*	AML	GENR	*UserDef-Name*	*Index*	*Value*	-	-
SOLVE	*MAT-ID*	AML	GENR	*UserDef-Name*	*Norm-Flag*	*Nu-mIter*	RTOL	CTOL
FIX	*MAT-ID*	AML	GENR	*UserDef-Name*	*Index*	-	-	-
EADD	*MAT-ID*	*ExpTyp*	*File*	*UserDef-Name*	*Direct-ory*	-	-	-
EDEL	*MAT-ID*	*ExpType / In-dex*	-	*UserDef-Name*	-	-	-	-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TBFT Specifications for Chaboche Kinematic Hardening Plasticity Models

TBFT (p. 1969), *Oper*, *MATID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*, –, –, *Option10*

Oper

The operation to perform:

Operation Set 1 (Curve-Fitting)

- - - - - - - - - - - - - - - - - - -

FCASE --

Define a case/constitutive model for plasticity.

FADD --

Define a constitutive model.

FDEL --

Delete a constitutive model.

FSET --

Write data related to a constitutive model to the database (same as **TB (p. 1899)** command).

SET --

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

CDEL --

Deletes coefficients at current reference temperature. Applicable only for temperature-dependent coefficients.

SOLVE --

Solve for coefficients.

FIX --

Fix (hold constant) the coefficient you specify in *Option4*.

Operation Set 2 (Experimental Data)

- - - - - - - - - - - - - - - - - - -

EADD --

Add experimental data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EDEL --

Delete experimental data.

Other Operations

- - - - - - - - - - - - - - - - - - -

LIST --

List all data associated with the material model represented by the material ID number (*MATID*).

MATID

Material reference identification number (same as *MAT* argument used in the **TB (p. 1899)** command). Valid value is any number greater than zero (default = 1) but less than 100,000.

Option1

For *Oper* = FCASE, set to either NEW or FINI. The command **TBFT (p. 1969)**,FCASE,*MATID*,NEW initializes a new curve-fitting case. (The **TBFT (p. 1969)**,FADD commands described next are always issued between **TBFT (p. 1969)**,FCASE,*MATID*,NEW and **TBFT (p. 1969)**,FCASE,*MATID*,FINI commands.) After issuing **TBFT (p. 1969)**,FCASE,*MATID*,FINI, the Chaboche model is created and is ready to be used to perform other curve-fitting operations. For more information, see Material Curve-Fitting in the *Material Reference*.

For *Oper* = FADD, set to PLAS to add options/parameters for the new case being created (via **TBFT (p. 1969)**,FCASE,*MATID*,NEW, which must be issued *before* the **TBFT (p. 1969)**,FADD command). This operation specifies the order of the Chaboche kinematic model.

For other curve-fitting function operations (*Oper* = FDEL, FSET, SET, CDEL, SOLVE or FIX), set to CASE.

For adding or deleting your experiment (*Oper* = EADD *or* EDEL), this option specifies the experiment type. The only valid option is UNIA (plastic strain vs. true stress).

Option2

For *Oper* = FCASE (defining your Chaboche case), set to CPLA.

For *Oper* = FDEL, FSET, SET, CDEL, SOLVE, or FIX (curve-fitting function operations), this value specifies the case name being operated on.

For *Oper* = FADD (specifying options for the plasticity model), valid options are:

CHAB -- Chaboche kinematic hardening (required for any defined curve-fitting case)

BISO -- Bilinear isotropic hardening (optional)

MISO -- Multilinear isotropic hardening (optional)

VOCE -- Nonlinear isotropic hardening, Voce model (optional)

The **TBFT (p. 1969)**,FADD command can be issued twice, once to specify the order of the Chaboche model, and again to specify the isotropic hardening option.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Only *one* of the options BISO, MISO or VOCE can be used in a single curve-fitting case, and none of those options are required.)

For *Oper* = EADD (specifying a file name from which to get experimental data), place that string here. A valid entry is any file name string. You can either:

- Enter the entire `path\filename.extension` string and leave the next two values (*Option3* and *Option4*) blank, *or*
- Specify the name here, the extension in the next value, and the path in the following value.

Option3

For *Oper* = FCASE, this value specifies the case name.

For *Oper* = FADD, this value specifies either:

- The order of the Chaboche kinematic hardening model (*Option2* = CHAB), *or*
- The number of terms in the MISO model (*Option2* = MISO).

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value contains the file extension.

Option4

When fixing a specific coefficient to a desired value (*Oper* = FIX), this value specifies the index of that coefficient. Valid values vary from 1 to *n*, where *n* is the total number of coefficients. Default = 1.

For *Oper* = SET, see Table 228: Set Operations (p. 1974).

You can also specify *TREF* to indicate the reference temperature, or *COMP* for a partial/complete solution (only for bulk, only for shear, or all coefficients).

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value contains the directory/path specification.

If *Oper* = SOLVE, this value specifies the normalized/non-normalized option. This option is not available for Chaboche curve-fitting.

Option5

For *Oper* = SET, refer to the following table.

Table 229: Set Operations

Purpose	Option4	Option5
Set the value of the coefficient	Index of coefficient	Value of coefficient
Set temperature dependency ON/OFF	TDEP	1 for ON and 0 for OFF
Set reference temperature	TREF	Temperature value

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *Oper* = SOLVE, use this value to specify the number of iterations to be used in the calculation of the coefficients. Valid value is any positive integer. Default = 1000.

If you are specifying a coefficient to be held constant (*Oper* = FIX):

1 – Fix the specified coefficient.

0 – Allow the specified coefficient to vary (disable fixing).

Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid value is 0.0 to 1.0. Default = 0.0.

Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid value is 0 to 1. Default = 0.

–

Reserved for future use.

–

Reserved for future use.

Option10

If *Oper* = SOLVE, enables parameter-scaling when set to 1. Default = 0. Used for Chaboche material curve-fitting.

This table summarizes the format for Chaboche curve-fitting operations via **TBFT (p. 1969)**:

Chaboche Model Command Summary

Oper	*MAT-ID*	*Option1*	*Option2*	*Option3*	*Option4*	*Option5*	*Option6*	*Option7*
FCASE	*MAT-ID*	NEW	CPLA	Case Name				
FCASE	*MAT-ID*	FINI						
FADD	*MAT-ID*	PLAS	Option	Order				
FDEL	*MAT-ID*	CASE	Case Name	NA				
FSET	*MAT-ID*	CASE	Case Name	NA				
SET	*MAT-ID*	CASE	Case Name	NA	Index	Value		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Oper	*MAT-ID*	*Op-tion1*	*Op-tion2*	*Op-tion3*	*Op-tion4*	*Op-tion5*	*Op-tion6*	*Op-tion7*
SOLVE	*MAT-ID*	CASE	Case Name	NA	*Not used*	Num Iter	RTOL	CTOL
FIX	*MAT-ID*	CASE	Case Name	NA	Index	Fix / Unfix Flag		
EADD	*MAT-ID*	Exp Type	File	Extension	Directory			
EDEL	*MAT-ID*	Exp Type / Index						

TBFT Specifications for Creep Models

TBFT (p. 1969), *Oper*, *MATID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*, –, –, *Option10*

Oper

The operation to perform:

Operation Set 1 (Curve-Fitting)

- - - - - - - - - - - - - - - - - - -

FADD --

Define a constitutive model.

FDEL --

Delete a constitutive model.

FSET --

Write data related to a constitutive model to the database (same as **TB (p. 1899)** command).

SET --

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

CDEL --

Deletes coefficients at current reference temperature. Applicable only for temperature dependent coefficients.

SOLVE --

Solve for coefficients.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FIX --

Fix (hold constant) the coefficient you specify in *Option4*.

Operation Set 2 (Experimental Data)

- - - - - - - - - - - - - - - - - - -

EADD --

Add experimental data.

EDEL --

Delete experimental data.

Other Operations

- - - - - - - - - - - - - - - - - - -

LIST --

List all data associated with the material model represented by the material ID number (*MATID*)

MATID

Material reference identification number (same as *MAT* argument used in the **TB (p. 1899)** command). Valid value is any number *n*, where 0 < *n* < 100,000. Default = 1.

Option1

For curve-fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE or FIX) this value specifies the category (CREEP).

For adding or deleting your experiment (*Oper* = EADD *or* EDEL), this value specifies the experimental data type (CREEP).

Option2

For curve-fit function operations (*Oper* = FADD, FDEL, FSET, SET, CDEL, SOLVE, or FIX), this value specifies constitutive model type. The valid values are listed in Table 231: Creep Options (p. 1983) below.

When you need to specify a file name from which to get experimental data (*Oper* = EADD), place that string here. Valid value is any file name string. You can enter the entire `path\filename.extension` string and leave the next two values (*Option3* and *Option4*) blank, or you can specify the name here, the extension in the next value, and the path following.

Option3

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value will contain the file extension.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option4

When you are working on a specific coefficient (*Oper* = FIX), this value, specifies the index of that coefficient. Valid values vary from 1 to *n*, where *n* is the total number of coefficients. Default = 1.

For *Oper* = SET, see Table 230: Set Operations (p. 1982), below.

If a file name for experimental data is being specified in *Option2* (*Oper* = EADD), this value will contain the directory/path specification.

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid values are 0 for non-normalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

Option5

If *Oper* = SOLVE, use this value to specify the number of iterations to be used in the calculation of the coefficients. Valid value is any positive integer. Default = 1000.

If you are specifying a coefficient to be held constant (*Oper* = FIX):

1 – Fix the specified coefficient.

0 – Allow the specified coefficient to vary (disable fixing).

For *Oper* = SET, see Table 230: Set Operations (p. 1982), below.

Table 230: Set Operations

Purpose	Option4	Option5
Set the value of the coefficient	Index of coefficient	Value of coefficient
Set temperature dependency ON[a]/OFF[b]	TDEP	1 -- ON 0 -- OFF
Set reference temperature	TREF	Temperature value

[a] When TDEP is OFF, the Arrhenius term (C4) in the strain-hardening creep equation is calculated. Because **TBTEMP (p. 1993)** is not included in the curve-fitting, the experimental data (in absolute temperature) must be provided via **/TEMP**.

[b] When TDEP is ON, the Arrhenius term is set to 0 and the constants are calculated separately for each temperature. The curve-fitting process uses **TBTEMP (p. 1993)**. You must provide the experimental data using the same units as specified via **TREF (p. 2013)**.

Option6

If *Oper* = SOLVE, specifies the allowed tolerance in residual change to stop an iteration. Valid value is 0.0 to 1.0. Default = 0.0.

Option7

If *Oper* = SOLVE, specifies the allowed tolerance in coefficient change to stop an iteration. Valid value is 0 to 1. Default = 0.

–

Reserved for future use.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

–

Reserved for future use.

Option10

If *Oper* = SOLVE, enables parameter-scaling when set to 1. Default = 0. Used for creep material curve-fitting.

Table 231: Creep Options

Category	Name	Option
CREEP	SHAR	NA
CREEP	THAR	NA
CREEP	GEXP	NA
CREEP	GGRA	NA
CREEP	GBLA	NA
CREEP	MTHA	NA
CREEP	MSHA	NA
CREEP	GGAR	NA
CREEP	EXPO	NA
CREEP	NORT	NA
CREEP	PSTH	NA
CREEP	PSRP	NA
CREEP	GTHA	NA

This table summarizes the format for creep curve-fitting operations via **TBFT (p. 1969)**:

Creep Model Command Summary

Oper	***MAT ID***	***Option1***	***Option2***	***Option3***	***Option4***	***Option5***	***Option6***	***Option7***
FADD	*MAT ID*	CREEP	Option	NA				
FDEL	*MAT ID*	CREEP	Option	NA				
FSET	*MAT ID*	CREEP	Option	NA				
SET	*MAT ID*	CREEP	Option	NA	Index	Value		
SOLVE	*MAT ID*	CREEP	Option	NA	Norm Flag	Num Iter	RTOL	CTOL
FIX	*MAT ID*	CREEP	Option	NA	Index	Fix / Unfix Flag		

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Oper	*MAT-ID*	*Op-tion1*	*Op-tion2*	*Op-tion3*	*Op-tion4*	*Op-tion5*	*Op-tion6*	*Op-tion7*
EADD	*MAT-ID*	Exp Type	File	Extension	Directory			
EDEL	*MAT-ID*	Exp Type / Index						

TBFT Specifications for Thermomechanical Fatigue (TMF) and Plasticity Model Combinations

TBFT (p. 1969), *Oper*, *MATID*, *Option1*, *Option2*, *Option3*, *Option4*, *Option5*, *Option6*, *Option7*

Oper

The operation to perform:

Operation Set 1 (Curve-Fitting)

- - - - - - - - - - - - - - - - - - -

FADD --

Define a constitutive model.

FDEL --

Delete a constitutive model.

FSET --

Write data related to a constitutive model to the database (same as **TB (p. 1899)** command).

SET --

Initialize coefficients of a constitutive model for nonlinear curve-fitting procedure.

CDEL --

Deletes coefficients at current reference temperature. Applicable only for temperature-dependent coefficients.

AINI --

Automatically initialize coefficients based on elastic properties and experimental data. (See "Notes" (p. 1987).)

SOLVE --

Solve for coefficients.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PSOLVE --

Custom multistep solve for coefficients.

FIX --

Fix (hold constant) the coefficient you specify in *Option4*.

Operation Set 2 (Experimental Data)

- - - - - - - - - - - - - - - - - - -

EADD --

Add experimental data.

EDEL --

Delete experimental data.

Other Operations

- - - - - - - - - - - - - - - - - - -

LIST --

List all data associated with the material model represented by the material ID number (*MATID*).

PSCA

Activate parameter-scaling to improve optimization performance. Valid for the UserMat curve-fitting method only (**TB (p. 1899)**,USER,,,,MXUP).

MATID

Material reference identification number. (Same as **TB (p. 1899)**,,*MATID*.) Valid value is any number *n*, where 0 < *n* < 100,000. Default = 1.

Option1

AML – For curve-fitting function operations (*Oper* = Operation Set 1), this value specifies the category.

UNIA – For adding or deleting your experiment (Operation Set 2), this value specifies the experimental data type.

Option2

For curve-fitting function operations (*Oper* = Operation Set 1), this value specifies the constitutive model type. The only valid value is GENR (generic).

To obtain experimental data (*Oper* = EADD in Operation Set 2) from a file, specify any valid file name. (You can either specify the entire *path**filename*.*extension* string here and leave *Option3* and *Option4* blank, or specify *filename* here, *extension* in *Option3*, and *path* in *Option4*.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option3

For curve-fitting function operations (*Oper* = Operation Set 1), specify a unique name for your curve-fitting model.

For obtaining experimental data (*Oper* = EADD in Operation Set 2) from a file specified in *Option2*, specify the file extension.

Option4

When fixing a specific coefficient to a desired value (*Oper* = FIX), this value specifies the index of that coefficient. Valid values vary from 1 to *n*, where *n* is the total number of coefficients. Default = 1.

For *Oper* = SET, see Table 232: Set Operations (p. 1986).

You can also specify *TREF* to indicate the reference temperature, or *COMP* for a partial/complete solution (only for bulk, only for shear, or all coefficients).

If *Oper* = SOLVE, this value specifies the curve-fitting procedure. Valid values are 0 for non-normalized least squares curve-fitting procedure, and 1 for normalized least squares curve-fitting procedure.

For obtaining experimental data (*Oper* = EADD) from a file specified in *Option2*, specify the path in which the file resides.

Option5

For *Oper* = SET, refer to the following table:

Table 232: Set Operations

Purpose	Option4	Option5
Set the value of the coefficient	Index of coefficient	Value of coefficient
Set temperature dependency ON/OFF	TDEP	1 -- ON 0 -- OFF
Set reference temperature	TREF	Temperature value

If *Oper* = PSCA, use this value to specify the scaling option to be used in the parameter-fitting process. Options are LINE (default), EXPO (exponential), SQRD (squared) or LOG (logarthmic). Use with the UserMat curve-fitting method.

If *Oper* = SOLVE, use this value to specify the number of iterations to be used in the calculation of the coefficients. Valid value is any positive integer. Default = 1000.

If you are specifying a coefficient to be held constant (*Oper* = FIX):

1 – Fix the specified coefficient.

0 – Allow the specified coefficient to vary (disable fixing).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option6

If *Oper* = SOLVE, specify the allowed tolerance in residual change to stop an iteration. Valid value is 0.0 to 1.0. Default = 0.0.

Option7

If *Oper* = SOLVE, specify the allowed tolerance in coefficient change to stop an iteration. Valid value is 0 to 1. Default = 0.

Supported Material Models

Material Model	TB Command[a]
Elastic Isotropic (p. 1922)	**TB (p. 1899)**,ELASTIC
Isotropic Hardening (p. 1941)	**TB (p. 1899)**,PLASTIC,,,,BISO
Kinematic Hardening (p. 1913)	**TB (p. 1899)**,CHABOCHE
Rate-Dependent Plasticity (p. 1945)	**TB (p. 1899)**,RATE
Kinematic Static Recovery (p. 1941)	**TB (p. 1899)**,PLASTIC,,,,KSR2
Isotropic Static Recovery (p. 1941)	**TB (p. 1899)**,PLASTIC,,,,ISR

[a] Issue **TB (p. 1899)** to define the material model *before* issuing **TBFT (p. 1969)**,FADD.

Notes

The **TBFT (p. 1969)** command provides tools for comparing experimental material data to the program-provided calculated data for various nonlinear material options. Based on curve-fitting comparisons and error norms, choose the model to use during the solution phase of the analysis according to the best fit. All of the capabilities of the **TBFT (p. 1969)** command are accessible interactively via the standard material GUI. For more information, see Material Curve-Fitting in the *Material Reference*.

Display material model data associated with both the **TB (p. 1899)** command and the **TBFT (p. 1969)**,FSET command via **TBLIST (p. 1988)**,ALL,ALL.

Material model data associated with the *most recent* **TB (p. 1899)** or **TBFT (p. 1969)**,FSET command overwrites previous data.

Display material model data associated with both the **TB (p. 1899)** command and the **TBFT (p. 1969)**,FSET command via **TBLIST (p. 1988)**,ALL,ALL.

The capability to fix coefficients (*Option4* = FIX) applies only to nonlinear curve-fitting (as listed in Table 6.4: Valid Hyperelastic Curve-Fitting Model Types.

The uniaxial, biaxial, and shear experimental data use *engineering* stress. The volumetric data uses *true* stress. See the *Material Reference* for details about experimental data for creep and viscoelasticity.

The hyperelastic option AML,GENR is a generalized framework where the parameters (defined *prior* to issuing **TBFT (p. 1969)**) are imported directly from **TB (p. 1899)** and **TBDATA (p. 1960)**. Parameter-fitting uses this framework for the thermomechanical fatigue, geomechanical, and TNM material models.

TBFT (p. 1969),AINI (p. 1984) calls a Python script that predicts initial coefficient values for some hyperelasticity material models. To ensure that the necessary Python libraries are installed, Mechanical APDL in-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

stallation includes a Python version that has been tested and known to work with this feature. Therefore, you cannot use the feature with any other installed version of Python. Delete the PYTHONHOME environment variable if it exists.

TBFT (p. 1969) does not support saving to (**SAVE (p. 1645)**) and resuming from (**RESUME (p. 1601)**) the database file. You must therefore rerun the curve-fitting analysis and then replot (**TBFPLOT (p. 1968)**).

Menu Paths

Main Menu> Preprocessor> Material Props> Material Models

TBLE

Specifies "Data table properties" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and will appear in the log file (`Jobname`.LOG) if status is requested for some items under **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

Utility Menu>List>Status>Preprocessor>Data Tables

TBLIST, *Lab*, *MAT*

Lists the material data tables.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Data table label. (See the **TB (p. 1899)** command for valid labels.) Defaults to the active table. If ALL, list data for all labels.

MAT

Material number to be listed (defaults to the active material). If ALL, list data tables for all materials.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

This command is a utility command, valid anywhere.

Menu Paths

Utility Menu>List>Properties>Data Tables

TBMODIF, *ROW*, *COL*, *VALUE*

Modifies data for the material data table (GUI).

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ROW, COL

The row and column numbers of the table entry to be modified.

VALUE

The new value to be used in the *ROW*, *COL* location.

Notes

The **TBMODIF (p. 1989)** command modifies data for the table specified on the last **TB (p. 1899)** command.

For temperature-dependent data, the command uses the temperature specified via the last **TBTEMP (p. 1993)** command.

The command is generated by the program's Graphical User Interface (GUI). It appears in the log file (`Jobname`.LOG) if a **TB (p. 1899)** material data table is graphically edited in spreadsheet fashion. The command is *not* intended to be typed in directly during an analysis session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

The command supports the following material data tables (**TB (p. 1899)**,*Lab* values):

ANEL (p. 1907) – Anisotropic elasticity
AVIS (p. 1908) – Anisotropic viscosity
CFOAM (p. 1912) – Crushable foam
DLST (p. 1919) – Anisotropic dielectric loss tangent
DPER (p. 1920) – Anisotropic electric permittivity
ELST (p. 1923) – Anisotropic elastic loss tangent
FCON (p. 1924) – Fluid conductance data
GASKET (p. 1927) – Gasket
GURSON (p. 1928) – Gurson pressure-dependent plasticity
HFLM (p. 1928) – Film coefficient data

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HILL (p. 1929) – Hill anisotropy

JOIN (p. 1932) – Joint

MIGR (p. 1938) – Migration

NLISO (p. 1939) – Voce isotropic hardening law

PIEZ (p. 1941) – Piezoelectric matrix

PLASTIC (p. 1941) (*TB0PT* = BISO) – Bilinear isotropic hardening

PLASTIC (p. 1941) (*TB0PT* = BKIN) – Bilinear kinematic hardening

PRONY (p. 1943) – Prony series

PZRS (p. 1944) – Piezoresistivity

SHIFT (p. 1946) – Shift function for viscoelastic materials

SMA (p. 1948) – Shape memory alloy

STATE (p. 1949) – User-defined state variables

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBPLOT, *Lab*, *MAT*, *TB0PT*, *TEMP*, *SEGN*

Displays the material data table.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Data table label. Valid labels are: MELAS, BKIN, BISO, BH, GASKET, and JOIN. Defaults to the active table label. For B-H data, also valid are: NB to display NU-B^2, MH to display MU vs. H, and SBH, SNB, SMH to display the slopes of the corresponding data.

MAT

Material number to be displayed (defaults to the active material).

TB0PT

Gasket material or joint element material option to be plotted.

The following gasket material options are valid only when *Lab* = GASKET:

ALL

Plots all gasket data.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COMP

Plots gasket compression data only.

LUNL

Plots gasket linear unloading data with compression curve.

NUNL

Plots gasket nonlinear unloading data only.

The following joint element material options are valid only when *Lab* = JOIN:

JNSA

Plots nonlinear stiffness data that is applicable to all relevant directions.

JNS*n*

Plots only the specified nonlinear stiffness data. The "*n*" can be 1, 4, or 6. For example, JNS4 plots only the nonlinear stiffness data specified in the local direction 4 (ROTX).

JNDA

Plots nonlinear damping data that is applicable to all relevant directions.

JND*n*

Plots only the specified nonlinear damping data. The "*n*" can be 1, 4, or 6. For example, JND4 plots only the nonlinear damping data specified in the local direction 4 (ROTX).

JNFA

Plots nonlinear hysteretic friction data that is applicable to all relevant directions.

JNF*n*

Plots only the specified nonlinear hysteretic friction data. The "*n*" can be 1, 4, or 6. For example, JNF4 plots only the nonlinear hysteretic friction data specified in local direction 4 (ROTX).

TEMP

Specific temperature at which gasket data or joint element material data will be plotted (used only when *Lab* = GASKET or JOIN). Use *TEMP* = ALL to plot gasket data or joint element material data at all temperatures.

SEGN

Segment number of plotted curve (valid only when *Lab* = GASKET):

NO

Segment number is not added to plotted curve (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

YES

Segment number is added to plotted curve. This option is ignored if the number of data points in a curve exceeds 20.

Notes

Only data for stress-strain, B-H, gasket curves, or joint element nonlinear material model curves can be displayed.

The *TBOPT* and *TEMP* values are valid only when *Lab* = GASKET or JOIN.

The *SEGN* value is valid only when *Lab* = GASKET.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Data Tables

TBPT,*Oper*,*X1*,*X2*,*X3*,...,*XN*

Defines a point on a nonlinear data curve.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Oper

Operation to perform:

DEFI

Defines a new data point (default). The point is inserted into the table in ascending order of *X1*. If a point already exists with the same *X1* value, it is replaced.

DELE

Deletes an existing point. The *X1* value must match the *X1* value of the point to be deleted (*XN* is ignored).

X1, X2, ..., XN

The N components of the point. N depends on the type of data table. Except for **TB (p. 1899)**,EXPE all other **TB (p. 1899)** Tables support only 2 components.

Notes

TBPT (p. 1992) defines a point on a nonlinear data curve (such as a stress-strain curve, B-H curve, etc.) at the temperature specified on the last **TBTEMP (p. 1993)** command. The meaning of the values depends on the type of data table specified on the last **TB (p. 1899)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

TBTEMP, *TEMP*, *KMOD*

Defines a temperature for a material data table.

PREP7 (p. 22): Data Tables (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TEMP

Temperature value (defaults to 0.0 if *KMOD* is blank).

KMOD

If blank, *TEMP* defines a new temperature. (Issue **TBLIST (p. 1988)** to list temperatures and data.)

Notes

The **TBTEMP (p. 1993)** command defines a temperature to be associated with the data on subsequent **TBPT (p. 1992)** or **TBDATA (p. 1960)** commands.

The defined temperature remains active until the next **TBTEMP (p. 1993)** command is issued.

Data values must be defined with the temperatures in ascending order.

This command is also valid in SOLUTION.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TCHG, *ENAME1*, *ENAME2*, *ETYPE2*

Converts 20-node degenerate tetrahedral elements to their 10-node non-degenerate counterparts.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ENAME1

Name (or the number) of the 20-node tetrahedron element that you want to convert. This argument is required.

ENAME2

Name (or the number) of the 10-node tetrahedron element to which you want to convert the *ENAME* elements. This argument is required.

ETYPE2

Element TYPE reference number for *ENAME2*. If *ETYPE2* is 0 or is not specified, the program selects the element TYPE reference number for *ENAME2*. See the "Notes" (p. 1994) section for details. This argument is optional.

Notes

The **TCHG (p. 1994)** command allows you to specify conversion of *any* selected 20-node brick that is degenerated into a tetrahedron to a 10-node tetrahedron.

The **TCHG (p. 1994)** command is useful when used in with the **MOPT (p. 1151)**,PYRA command. Twenty-node pyramid shaped elements may be used in the same volume with 10-node tetrahedra.

Performing a conversion is likely to create circumstances in which more than one element type is defined for a single volume.

If specified, *ETYPE2* will usually be the same as the local element TYPE number (**ET (p. 686)**,*ITYPE*) that was assigned to *ENAME2* with the **ET (p. 686)** command. You can specify a unique number for *ETYPE2* if you prefer. Although *ETYPE2* is optional, it may be useful when two or more *ITYPE*s have been assigned to the same element (for example, if two SOLID187 elements have been established in the element attribute tables for the current model, use the *ETYPE2* argument to distinguish between them). If *ETYPE2* is nonzero and it has *not* already been assigned to an element via **ET (p. 686)**, the program assigns the *ETYPE2* value to *ENAME2* as its element TYPE reference number.

If *ETYPE2* is 0 or is not specified, the program determines the element TYPE reference number for *ENAME2* in one of these ways:

- If *ETYPE2* is 0 or is not specified, and *ENAME2 does not appear* in the element attribute tables, the program uses the next available (unused) location in the element attribute tables to determine the element TYPE reference number for *ENAME2*.
- If *ETYPE2* is 0 or is not specified, and *ENAME2 appears* in the element attribute tables, the program uses *ENAME2*'s existing element TYPE reference number for *ETYPE2*. (If there is more

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

than one occurrence of *ENAME2* in the element attribute tables (each with its own TYPE reference number), the program uses the first *ENAME2* reference number for *ETYPE2* .)

You cannot use element conversion if boundary conditions or loads are applied directly to any selected elements.

For more information about converting degenerate tetrahedral elements, see Meshing Your Solid Model in the *Modeling and Meshing Guide*

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Change Tets

/**TEE**, *Label*, *Fname*, *Ext*, - -

Writes a list of commands to a specified file at the same time that the commands are being executed.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Label

Specifies how Mechanical APDL is to interpret this **/TEE (p. 1995)** command:

NEW --

Signals the beginning of the command text that is to be written to *Fname*. If *Fname* already exists, specifying NEW causes the contents of *Fname* to be overwritten.

APPEND --

Indicates that you want to append to *Fname* the command text that follows.

END --

Signals the end of the command text that is to be written to or appended to *Fname*.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

If you plan to execute the file as if it were a Mechanical APDL command, use the extension *.mac*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- -

Unused field.

Notes

You can use the **/TEE (p. 1995)** command to record a macro to a specified file at the same time that the macro is being executed. It is similar to the Linux **tee** command.

For more information about the **/TEE (p. 1995)** command, see the Introducing APDL of the *Ansys Parametric Design Language Guide*.

The following example illustrates the use of the **/TEE (p. 1995)** command. If you issue these commands:

```
/tee,new,myfile,mac
et,1,42,0,0,1
ex,1,3e7
/tee,end
/tee,append,myfile,mac
n,1,8
n,5,11
fill
ngen,5,5,1,5,1,0,1
/tee,end
```

the content of **myfile.mac** is:

```
et,1,42,0,0,1
ex,1,3e7
n,1,8
n,5,11
fill
ngen,5,5,1,5,1,0,1
```

This command is valid in any processor, but only during an interactive run.

THEXPAND, *KEY*

Enables or disables thermal loading

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Activation key:

ON

Thermal loading is included in the load vector (default).

OFF

Thermal loading is not included in the load vector.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Temperatures applied in the analysis are used by default to evaluate material properties and contribute to the load vector if the temperature does not equal the reference temperature and a coefficient of thermal expansion is specified.

Use **THEXPAND (p. 1996)**,OFF to evaluate the material properties but not contribute to the load vector. This capability is particularly useful when performing a harmonic analysis where you do not want to include harmonically varying thermal loads. It is also useful in a modal analysis when computing a modal load vector but excluding the thermal load.

This command is valid for all analysis types except linear perturbation modal and linear perturbation harmonic analyses. For these two linear perturbation analysis types, the program internally sets **THEXPAND (p. 1996)**,OFF, and it cannot be set to ON by using this command (**THEXPAND (p. 1996)**,ON is ignored).

Menu Paths

This command cannot be accessed from a menu.

THOPT,*Refopt*,*REFORMTOL*,*NTABPOINTS*,*TEMPMIN*,*TEMPMAX*,--,*ALGO*

Specifies nonlinear transient thermal solution options.

SOLUTION (p. 38): Analysis Options (p. 38)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Refopt

Matrix reform option.

FULL

Use the full Newton-Raphson solution option (default). All subsequent input values are ignored.

QUASI

Use a selective reform solution option based on *REFORMTOL*.

REFORMTOL

Property change tolerance for Matrix Reformation (.05 default). The thermal matrices are reformed if the maximum material property change in an element (from the previous reform time) is greater than the reform tolerance. Valid only when *Refopt* = QUASI.

NTABPOINTS

Number of points in Fast Material Table (64 default). Valid only when *Refopt* = QUASI.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TEMPMIN

Minimum temperature for Fast Material Table. Defaults to the minimum temperature defined by the **MPTEMP (p. 1178)** command for any material property defined. Valid only when *Refopt* = QUASI.

TEMPMAX

Maximum temperature for Fast Material Table. Defaults to the maximum temperature defined by the **MPTEMP (p. 1178)** command for any material property defined. Valid only when *Refopt* = QUASI.

--

Reserved field.

ALGO

Specifies which solution algorithm to apply:

0

Multipass (default).

1

Iterative.

Valid only when *Refopt* = QUASI.

Notes

The QUASI matrix reform option is supported by the ICCG, JCG, PCG, and sparse solvers only (**EQSLV (p. 648)**). The Quasi method is an approximation to the FULL method and will not be as accurate when the nonlinearity is strong. However, you can control the inaccuracy by using small time steps.

For *Refopt* = QUASI:

- Results from a restart may be different than results from a single run because the stiffness matrices are always recreated in a restart run, but may or may not be in a single run (depending on the behavior resulting from the *REFORMTOL* setting). Additionally, results may differ between two single runs as well, if the matrices are reformed as a result of the *REFORMTOL* setting.

For more information, see Solution Algorithms Used in Transient Thermal Analysis in the *Thermal Analysis Guide*.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>Analysis Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TIFF, *Keyword*, *OPT*

Provides TIFF file export for Mechanical APDL displays.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Keyword

Specifies various TIFF file export options.

COMP

If *Keyword* = COMP, then *OPT* controls data compression for the output file. If COMP = 0, then compression is off. If COMP = 1 (default), then compression is on.

ORIENT

If *Keyword* = ORIENT, then *OPT* will determine the orientation of the entire plot. *OPT* can be either Horizontal (default) or Vertical.

COLOR

If *Keyword* = COLOR, then *OPT* will determine the color attribute of the saved file. *OPT* can be 0, 1, or 2, corresponding to Black and White, Grayscale, and Color (default), respectively.

TMOD

If *Keyword* = TMOD, then *OPT* will determine the text method. *OPT* can be either 1 or 0, corresponding to bitmap text (default) or line stroke text, respectively.

DEFAULT

If *Keyword* = DEFAULT, then all of the default values, for all of the Keyword parameters listed above, are active.

OPT

OPT can have the following names or values, depending on the value for *Keyword* (see above).

1 or 0

If *Keyword* = COMP, a value or 1 (on) or 0 (off) will control compression for the TIFF file.

Horizontal, Vertical

If *Keyword* = ORIENT, the terms Horizontal or Vertical determine the orientation of the plot.

0, 1, 2

If *Keyword* = COLOR, the numbers 0, 1, and 2 correspond to Black and White, Grayscale and Color, respectively.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1, 0

If *Keyword* = TMOD, the values 1 and 0 determine whether bitmap (1) or stroke text (0) fonts will be used

Menu Paths

Utility Menu>PlotCtrls>HardCopy>ToFile

TIME, *TIME*

Sets the time for a load step.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TIME

Time at the end of the load step.

Command Default

Previous *TIME* + 1.0 (at each load step), corresponding to the load step number.

Notes

Associates the boundary conditions at the end of the load step with a particular *TIME* value.

TIME must be a positive, nonzero, monotonically increasing quantity that "tracks" the input history. Units of time should be consistent with those used elsewhere (for properties, creep equations, etc.).

Typically, for the first load step *TIME* defaults to 1. However, for the first load step of a mode-superposition transient analysis (**ANTYPE (p. 162)**,TRANS *and* **TRNOPT (p. 2017)**,MSUP), the **TIME (p. 2000)** command is ignored and a static solution is performed at *TIME* = 0.

For a full transient analyses, the command's default behavior does not apply. You must specify a time for each load step and it must be greater than the time at the end of the prior load step.

TIME does not apply to modal (**ANTYPE (p. 162)**,MODAL), harmonic (**ANTYPE (p. 162)**,HARMIC), or substructure (**ANTYPE (p. 162)**,SUBSTR) analyses.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Basic

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substeps

Main Menu>Solution>Analysis Type>Sol'n Controls>Basic

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substeps

---

TIMERANGE, *TMIN*, *TMAX*

Specifies the time range for which data are to be stored.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TMIN

Minimum time (defaults to first time (or frequency) point on the file).

TMAX

Maximum time (defaults to last time (or frequency) point on the file).

Command Default

Include all time (or frequency) points in the range.

Notes

Defines the time (or frequency) range for which data are to be read from the file and stored in memory. Use the **NSTORE (p. 1287)** command to define the time increment.

Use **PRTIME (p. 1522)** or **PLTIME (p. 1441)** to specify the time (frequency) range for cyclic mode-superposition harmonic analyses.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

TIMINT, *Key*, *Lab*

Turns on transient effects.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Key

Transient effects key:

OFF

No transient effects (static or steady-state).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ON

Include transient (mass or inertia) effects.

Lab

Degree of freedom label:

ALL

Apply this key to all appropriate labels (default).

STRUC

Apply this key to structural DOFs.

THERM

Apply this key to thermal DOFs.

ELECT

Apply this key to electric DOFs.

MAG

Apply this key to magnetic DOFs.

FLUID

Apply this key to fluid DOFs.

DIFFU

Apply this key to concentration of DOFs.

Command Default

Include transient effects (ON) (**ANTYPE (p. 162)**,TRANS).

Notes

Indicates whether this load step in a full transient analysis should use time integration, that is, whether it includes transient effects (e.g. structural inertia, thermal capacitance) or whether it is a static (steady-state) load step for the indicated DOFs. Transient initial conditions are introduced at the load step having *Key* = ON. Initial conditions are then determined from the previous two substeps. Zero initial velocity and acceleration are assumed if no previous substeps exist. See the *Structural Analysis Guide*, the *Thermal Analysis Guide*, and the *Low-Frequency Electromagnetic Analysis Guide* for details.

This command is also valid in PREP7.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters

Main Menu>Solution>Analysis Type>Sol'n Controls>Transient

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters

---

TIMP, *ELEM*, *CHGBND*, *IMPLEVEL*

Improves the quality of tetrahedral elements that are not associated with a volume.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ELEM

Identifies the tetrahedral elements to be improved. Valid values are ALL and P. If *ELEM* = ALL (default), improve all selected tetrahedral elements. If *ELEM* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

CHGBND

Specifies whether to allow boundary modification. Boundary modification includes such things as changes in the connectivity of the element faces on the boundary and the addition of boundary nodes. (Also see the Notes section below for important usage information for *CHGBND*.)

0

Do not allow boundary modification.

1

Allow boundary modification (default).

IMPLEVEL

Identifies the level of improvement to be performed on the elements. (Improvement occurs primarily through the use of face swapping and node smoothing techniques.)

0

Perform the least amount of swapping/smoothing.

1

Perform an intermediate amount of swapping/smoothing.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2

Perform the greatest amount of swapping/smoothing.

3

Perform the greatest amount of swapping/smoothing, plus additional improvement techniques (default).

Notes

The **TIMP (p. 2003)** command enables you to improve a given tetrahedral mesh by reducing the number of poorly-shaped tetrahedral elements (in particular, the number of sliver tetrahedral elements)--as well as the overall number of elements--in the mesh. It also improves the overall quality of the mesh.

TIMP (p. 2003) is particularly useful for an imported tetrahedral mesh for which no geometry information is attached.

Regardless of the value of the *CHGBND* argument, boundary mid-nodes can be moved.

When loads or constraints have been placed on boundary nodes or mid-nodes, and boundary mid-nodes are later moved, the program issues a warning message to let you know that it will not update the loads or constraints.

No boundary modification is performed if shell or beam elements are present in the mesh, even when *CHGBND* = 1.

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Improve Tets>Detached Elems

---

TINTP, *GAMMA*, *ALPHA*, *DELTA*, *THETA*, *OSLM*, *TOL*, *OSLM2*, - - , *AVSMOOTH*, *ALPHAF*, *ALPHAM*

Defines transient integration parameters.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

GAMMA

Amplitude decay factor for 2nd order transient integration, for example, structural dynamics (used only if *ALPHA*, *DELTA*, *ALPHAF*, and *ALPHAM* are blank, see "Notes" (p. 2006)). Defaults to 0.005.

Alternatively, you can input the application type for the analysis using one of the following labels. In this case, the program automatically sets the transient dynamic solver algorithm and settings based on the intended application. For more information, see Transient Dynamic Analysis Settings Based on Application in the *Structural Analysis Guide*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IMPA

Impact application.

HISP

High speed dynamic application.

MOSP

Moderate speed dynamic application.

LOSP

Low speed dynamic application.

QUAS

Quasi-static application.

ALPHA

2nd order transient integration parameter (used only if *GAMMA* is blank). Defaults to 0.2525.

DELTA

2nd order transient integration parameter (used only if *GAMMA* is blank). Defaults to 0.5050.

THETA

1st order transient (for example, thermal transient) integration parameter. Defaults to 1.0.

OSLM

Specifies the oscillation limit criterion for automatic time stepping of 1st order transients (for example, thermal transients). Defaults to 0.5 with a tolerance of *TOL*.

TOL

Tolerance applied to *OSLM*. Defaults to 0.0.

OSLM2

Specifies the oscillation limit for automatic time stepping of second order transients (for example, structural transients). The automatic time stepping algorithm will adjust to have approximately the specified number of points per cycle. You can specify a negative number (for example, -1) to define a time increment that is independent of the oscillation limit.

- -

Unused field.

AVSMOOTH

Smoothing flag option:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Include smoothing of the velocity (1st order system) or the acceleration (2nd order system) (default).

1

Do not include smoothing.

ALPHAF

Interpolation factor in HHT algorithm for force and damping terms (used only if *GAMMA* is blank). Defaults to 0.005.

ALPHAM

Interpolation factor in HHT algorithm for inertial term (used only if *GAMMA* is blank). Defaults to 0.0.

Command Default

GAMMA defaults to 0.005, which is the same as choosing the High Speed application (*GAMMA* = HISP). All other fields default as described above.

Notes

Used to define the transient integration parameters. For more information on transient integration parameters, refer to Transient Analysis in the *Theory Reference*.

For a full structural transient analysis, you may choose between the Newmark and HHT time integration methods (see the **TRNOPT (p. 2017)** command). In this case, if *GAMMA* is input and the integration parameters *ALPHA*, *DELTA*, *ALPHAF*, and *ALPHAM* are left blank, the program calculates the integration parameters, according to Equation 15.14 and Equation 15.18. Alternatively, you can input these integration parameters directly on this command. However, for the unconditional stability and second order accuracy of the time integration, these parameters should satisfy a specific relationship, as described in Description of Structural and Other Second Order Systems of the *Mechanical APDL Theory Reference*.

For mode-superposition transient analysis, the integration parameters *ALPHA* and *DELTA* are always calculated by the program from the *GAMMA* value, according to Equation 15.14. If an *ALPHA*, *DELTA* pair has been specified with *GAMMA* left blank, the input values are ignored, and integration parameters *ALPHA*, *DELTA* are calculated with *GAMMA* = 0.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Solution>Analysis Type>Sol'n Controls>Transient

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Amplitude Decay

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time Integration>Newmark Parameters

---

/TITLE, *Title*

Defines a main title.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Title

Input up to 72 alphanumeric characters. Parameter substitution may be forced within the title by enclosing the parameter name or parametric expression within percent (%) signs.

Notes

The title is carried through the printout and written on various files. The title written to a file is the title defined at that time. Special characters may be used within the title text. Subtitles may also be defined (**/STITLE (p. 1867)**).

This command is valid in any processor.

Menu Paths

Utility Menu>File>Change Title

/TLABEL, *XLOC*, *YLOC*, *Text*

Creates annotation text (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XLOC

Text *X* starting location (-1.0 < *X* < 1.6).

YLOC

Text Y starting location (-1.0 < *Y* < 1.0).

Text

Text string (60 characters maximum). Parameter substitution may be forced within the text by enclosing the parameter name or parametric expression within percent (%) signs.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Defines annotation text to be written directly onto the display at a specified location. This command is generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if annotation is used.

The command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

All text is shown on subsequent displays unless the annotation is disabled or deleted. Issue **/TSPEC (p. 2024)** to set the attributes of the text.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

TOFFST, *VALUE*

Specifies the temperature offset from absolute zero to zero.

SOLUTION (p. 38): Analysis Options (p. 38)
AUX12 (p. 62): General Radiation (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VALUE

Degrees between absolute zero and zero of temperature system used (should be positive).

Notes

Specifies the difference (in degrees) between absolute zero and the zero of the temperature system used. Absolute temperature values are required in evaluating certain expressions, such as for creep, swelling, radiation heat transfer, MASS71, etc. (The offset temperature is not used in evaluating emissivity.) Examples are 460° for the Fahrenheit system and 273° for the Celsius system. The offset temperature is internally included in the element calculations and does not affect the temperature input or output. If used in SOLUTION, this command is valid only within the first load step.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Temperature Units

Main Menu>Preprocessor>Material Props>Temperature Units

Main Menu>Preprocessor>Radiation Opts>Solution Opt

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Radiation Opt>Radiosity Meth>Solution Opt

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Temperature Units

Main Menu>Solution>Radiation Opts>Solution Opt

***TOPER**,*ParR*,*Par1*,*Oper*,*Par2*,*FACT1*,*FACT2*,*CON1*

Operates on table parameters.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

Name of the resulting table parameter. The command will create a table array parameter with this name. Any existing parameter with this name will be overwritten.

Par1

Name of the first table parameter.

Oper

The operation to be performed: ADD. The operation is: ParR(i,j,k) = FACT1*Par1(i,j,k) + FACT2 *Par2(i,j,k) +CON1

Par2

Name of the second table parameter.

FACT1

The first table parameter multiplying constant. Defaults to 1.

FACT2

The second table parameter multiplying constant. Defaults to 1.

CON1

The constant increment for offset. Defaults to 0.

Notes

***TOPER (p. 2009)** operates on table parameters according to: ParR(i,j,k) = FACT1*Par1(i,j,k) + FACT2 *Par2(i,j,k) +CON1

Par1 and Par2 must have the same dimensions and the same variable names corresponding to those dimensions. Par1 and Par2 must also have identical index values for rows, columns, etc.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want a local coordinate system for the resulting array, you must dimension it as such using the ***DIM (p. 530)** command before issuing ***TOPER (p. 2009)**.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Table Operations

TORUS, *RAD1*, *RAD2*, *RAD3*, *THETA1*, *THETA2*

Creates a toroidal volume.

PREP7 (p. 22): Primitives (p. 25)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RAD1, RAD2, RAD3

Three values that define the radii of the torus. You can specify the radii in any order. The smallest of the values is the inner minor radius, the intermediate value is the outer minor radius, and the largest value is the major radius. (There is one exception regarding the order of the radii values--if you want to create a solid torus, specify zero or blank for the inner minor radius, in which case the zero or blank *must* occupy either the *RAD1* or *RAD2* position.) At least two of the values that you specify must be positive values; they will be used to define the outer minor radius and the major radius. See the diagram in the Notes section for a view of a toroidal sector showing all radii.

THETA1, THETA2

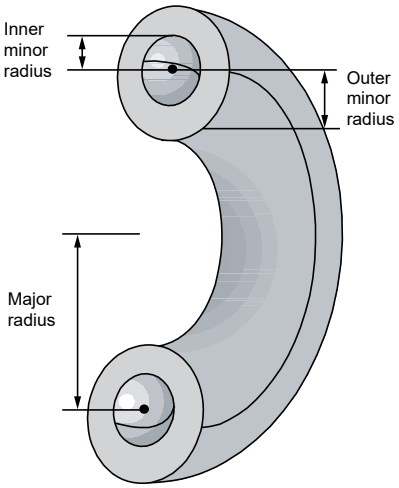
Starting and ending angles (either order) of the torus. Used for creating a toroidal sector. The sector begins at the algebraically smaller angle, extends in a positive angular direction, and ends at the larger angle. The starting angle defaults to 0° and the ending angle defaults to 360°.

Notes

Defines a toroidal volume centered about the working plane origin. A solid torus of 360° will be defined with four areas, each area spanning 180° around the major and minor circumference.

To create the toroidal sector shown below, the command **TORUS (p. 2010)**,5,1,2,0,180 was issued. Since "1" was the smallest radii value specified, it defined the inner minor radius; since "2" was the intermediate radii value specified, it defined the outer minor radius; and since "5" was the largest radii value specified, it defined the major radius. The values "0" and "180" defined the starting and ending angles of the torus.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Torus

TRANSFER, *KCNTO*, *INC*, *NODE1*, *NODE2*, *NINC*

Transfers a pattern of nodes to another coordinate system.

PREP7 (p. 22): Nodes (p. 31)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCNTO

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system.

INC

Increment all nodes in the given pattern by *INC* to form the transferred node pattern.

NODE1*, *NODE2*, *NINC

Transfer nodes from pattern beginning with *NODE1* to *NODE2* (defaults to *NODE1*) in steps of *NINC* (defaults to 1). If *NODE1* = ALL, *NODE2* and *NINC* are ignored and the pattern is all selected nodes (**NSEL (p. 1266)**). If *NODE1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component may be substituted for *NODE1* (*NODE2* and *NINC* are ignored).

Notes

Transfers a pattern of nodes from one coordinate system to another. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. Coordinate values are interpreted in the active coordinate system and are transferred directly.

A model generated in one coordinate system may be transferred to another coordinate system. The user may define several coordinate systems (translated and rotated from each other), generate a model

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in one coordinate system, and then repeatedly transfer the model to other coordinate systems. The model may be generated in any type of coordinate system (Cartesian, cylindrical, etc.) and transferred to any other type of coordinate system. Coordinate values (X, Y, Z, or R, θ, Z, or etc.) of the model being transferred are interpreted in the *active* coordinate system type, regardless of how they were generated. Values are transferred directly and are interpreted according to the type of coordinate system being transferred to. For example, transferring from a Cartesian coordinate system to a cylindrical coordinate system (not recommended) would cause X = 2.0 and Y = 3.0 values to be directly interpreted as R = 2.0 and θ = 3.0 values, respectively.

This command is also valid in the **/MAP (p. 1107)** processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Nodes

---

***TREAD**,*Par*,*Fname*,*Ext*,--,*NSKIP*

Reads data from an external file into a table array parameter.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

Table array parameter name as defined by the ***DIM (p. 530)** command.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

File name has no default.

Ext

Filename extension (eight-character maximum).

Extension has no default.

--

Unused field.

NSKIP

Number of comment lines at the beginning of the file being read that will be skipped during the reading. Default = 0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Use this command to read in a table of data from an external file into a table array parameter. The external file may be created using a text editor or by an external application or program. To be used by ***TREAD (p. 2012)**, the external file's encoding format must be UTF-8, and the file must be in tab-delimited, blank-delimited, or comma-delimited format. The TABLE type array parameter must be defined before you can read in an external file. See ***DIM (p. 530)** for more information.

This command is not applicable to 4- or 5-D tables.

Menu Paths

Utility Menu>Parameters>Array Parameters>Read from File

TREF, *TREF*

Defines the reference temperature for thermal strain calculations.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TREF

Reference temperature for thermal expansion.

Note:

If the uniform temperature (**TUNIF (p. 2027)**) is not specified, it is also set to this value.

Command Default

Reference temperature is 0.0 degrees.

Notes

Defines the reference temperature for the thermal strain calculations in structural analyses. Thermal strains are given by α * (T - TREF), where α is the coefficient of thermal expansion. Input the strain via ALPX, ALPY, ALPZ (the secant or mean coefficient value), or CTEX, CTEY, CTEZ (the instantaneous coefficient value), or the thermal strain value (THSX, THSY, THSZ). T is the element temperature. If α is temperature-dependent, *TREF* should be in the range of temperatures you define using the **MPTEMP (p. 1178)** command.

Reference temperatures may also be input *per material* by specifying a value on the **MP (p. 1160)** material property command:

MP (p. 1160),REFT,*MAT*,*C0*.

Only a constant (non-temperature-dependent) value is valid. The value input on the **TREF (p. 2013)** command applies to all materials not having a specified material property definition.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To convert temperature-dependent secant coefficients of thermal expansion (SCTE) data (properties ALPX, ALPY, ALPZ) from the definition temperature to the reference temperature defined via a **TREF (p. 2013)** (or **MP (p. 1160)**,REFT) command, issue the **MPAMOD (p. 1166)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Settings>Reference Temp

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Reference Temp

Main Menu>Solution>Define Loads>Settings>Reference Temp

Main Menu>Solution>Load Step Opts>Other>Reference Temp

/TRIAD, *Lab*

Shows the global XYZ coordinate triad on displays.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Display triad as follows:

ORIG

Display triad at global origin (default).

OFF

Turn off triad display.

LBOT

Display triad in lower left screen corner.

RBOT

Display triad in lower right screen corner.

LTOP

Display triad in upper left screen corner.

RTOP

Display triad in upper right screen corner.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

For efficiency, Mechanical APDL maintains a single data structure (segment) which includes the triad as a 3D data object.

If a 3D device is involved (**/SHOW (p. 1785)**,3D) and the graphics are not being displayed as multi-plots, the triad location is determined by the view settings for window #1.

A request for triad display anywhere except for the origin may yield an improper display in windows 2 through 5.

The program displays the same segment in all windows. The view settings of each window constitute the only difference in the display in the active windows.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Window Controls>Reset Window Options

Utility Menu>PlotCtrls>Window Controls>Window Options

/**TRLCY**, *Lab*, *TLEVEL*, *N1*, *N2*, *NINC*

Specifies the level of translucency.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

Apply translucency level to the items specified by the following labels:

ELEM

Elements. Use *N1*, *N2*, *NINC* fields for element numbers.

AREA

Solid model areas. Use *N1*, *N2*, *NINC* fields for area numbers.

VOLU

Solid model volumes. Use *N1*, *N2*, *NINC* fields for volume numbers.

ISURF

Isosurfaces (surfaces of constant stress, etc., value). Translucency varies with result value, to a maximum of the specified translucency level.

CM

Component group. Use *N1* for component name, ignore *N2* and *NINC*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CURVE

Filled areas under curves of line graphs. Use *N1*, *N2*, *NINC* fields for curve numbers.

ZCAP

If **/TYPE (p. 2034)**,*WN*,ZCAP is the current display type, then **/TRLCY (p. 2015)**,ZCAP,*TLEVEL* will display the model in window *WN* with the portion of the model in front of the section plane displayed at the translucency level *TLEVEL*.

ON, OFF

Sets the specified translucency display on or off. All other fields are ignored.

TLEVEL

Translucency level: 0.0 (opaque) to 1.0 (transparent).

N1, N2, NINC

Used only with labels as noted above. Apply translucency level to *Lab* items numbered *N1* to *N2* (defaults to *N1*) in steps of *NINC* (defaults to 1). If *N1* is blank or ALL, apply specified translucency level to entire selected range. If *Lab* is CM, use component name for *N1* and ignore *N2* and *NINC*. A value of *N1* = P allows you to graphically pick elements, areas, and volumes. You can then assign translucency levels to the entities via the picker. The *Lab* and *TLEVEL* fields are ignored when translucency is applied by picking.

Command Default

Zero translucency (opaque) level.

Notes

Specifies the level of translucency for various items. Issue **/TRLCY (p. 2015)**,DEFA to reset the default (0) translucency levels. This command is valid only on selected 2D and 3D graphics devices; see in the *Basic Analysis Guide* for more information on applying translucency.

For 2D devices, the program displays only the visible faces of the items being displayed. The information behind the facing planes is not displayed. Issuing the **/SHRINK (p. 1794)** command will force the hardware to display information behind the translucent items.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Translucency

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TRNOPT,*Method*,*MAXMODE*,*InitialAcc*,*MINMODE*,*MCFwrite*,*TINTOPT*,*VAout*,*DMPSFreq*,*EngCalc*,*MCkey*

Specifies transient analysis options.

SOLUTION (p. 38): Dynamic Options (p. 40)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
Product Restrictions (p. 2019)

Method

Solution method for the transient analysis:

FULL

Full method (default).

MSUP

Mode-superposition method.

MAXMODE

Largest mode number to be used to calculate the response (for *Method* = MSUP). Defaults to the highest mode calculated in the preceding modal analysis.

InitialAcc

Key to activate calculation of initial acceleration:

(blank)

Initial accelerations are not calculated (default).

INIL

Calculate initial acceleration for a full transient analysis using the lumped mass matrix.

MINMODE

Smallest mode number to be used (for *Method* = MSUP). Defaults to 1.

MCFwrite

Modal coordinates output key to the Jobname.mcf file (valid only for the mode-superposition method). To control how Jobname.mcf is written, specify options on the **MCFOPT (p. 1116)** command.

NO

Modal coordinates are not written to Jobname.mcf.

YES

Modal coordinates are written to the text file Jobname.mcf (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TINTOPT

Time integration method for the transient analysis:

NMK or 0

Newmark algorithm (default).

HHT or 1

HHT algorithm (valid only for the full transient method).

VAout

Velocities and accelerations output key (valid only for mode-superposition transient analysis):

NO

No output of velocities and accelerations (default).

YES

Write velocities and accelerations to the reduced displacement file, `Jobname.rdsp`.

DMPSFreq

Average excitation frequency (Hz) for the calculation of equivalent viscous damping from structural damping input (**DMPSTR (p. 563)** and **MP (p. 1160)**,DMPS). See Damping for more details. Defaults to zero. If an excitation frequency is not specified, structural damping is ignored. If tabular excitation frequency data is provided in a full transient analysis (*DMPSFreqTab* on **DMPSTR (p. 563)**), it supersedes this value.

EngCalc

Additional element energies calculation key:

NO

Do not calculate additional element energies (default).

YES

Calculate damping energy and work done by external loads.

MCkey

Modal coordinates output key to the `.rdsp` file (valid only for the mode-superposition method):

AUTO

Writing depends on the modal analysis settings of the **MXPAND (p. 1203)** command (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

YES

Always write the modal coordinates to the file `Jobname.rdsp`. A subsequent expansion pass (**EXPASS (p. 714)**) is not supported.

Notes

Specifies transient analysis (**ANTYPE (p. 162)**,TRANS) options. If used in SOLUTION, this command is valid only within the first load step. Use the **TINTP (p. 2004)** command to set transient integration parameters.

To include residual vectors in your mode-superposition transient analysis (*Method* = MSUP), specify **RESVEC (p. 1603)**,ON.

For *Method* = MSUP, *MAXMODE* and *MINMODE* are ignored after a modal restart analysis where remote modal files usage (**MODDIR (p. 1134)**) and residual vector calculation (**RESVEC (p. 1603)**,ON) have been activated.

Method = MSUP is not available for ocean loading.

By default in a mode-superposition transient analysis, reaction force and other force output contains only static contributions. If you want to postprocess the velocities, accelerations, and derived results (*Lab* = TOTAL, DAMP, or INERT on the **FORCE (p. 772)** command), set *VAout* = YES to activate velocity and acceleration output.

The calculation of additional energies (*EngCalc* = YES) is valid only for the full solution method (*Method* = FULL). The `Jobname.ESAV` file is always saved in this case. The numerical integration for damping energy and work are consistent only if solution data are written to the database for every substep (**OUTRES (p. 1336)**,ALL,ALL, **OUTRES (p. 1336)**,ESOL,ALL, or **OUTRES (p. 1336)**,VENG, ALL). For more information, see Damping Energy and Work Done by External Loads in the *Mechanical APDL Theory Reference*.

This command is also valid in PREP7.

Product Restrictions

Additional product restrictions for the **TRNOPT (p. 2017)** command are shown in the table below.

Command Option Method	**Available Products**
FULL (DOF = Structural)	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
FULL (DOF = Temp)	**Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on
MSUP	Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Menu Paths

Main Menu>Preprocessor>Loads>Analysis Type>Analysis Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Main Menu>Preprocessor>Loads>Analysis Type>New Analysis

Main Menu>Preprocessor>Loads>Analysis Type>Sol'n Controls>Transient

Main Menu>Solution>Analysis Type>Analysis Options

Main Menu>Solution>Analysis Type>New Analysis

Main Menu>Solution>Analysis Type>Sol'n Controls>Transient

TRPDEL, *NTRP1*, *NTRP2*, *TRPINC*

Deletes charged particle trace points.

POST1 (p. 51): Trace Points (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NTRP1*, *NTRP2*, *TRPINC

Delete points from *NTRP1* to *NTRP2* (defaults to *NTRP1*) in steps of *TRPINC* (defaults to 1). If *NTRP1* = ALL, *NTRP2* and *TRPINC* are ignored and all trace points are deleted. If *NTRP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Notes

Deletes charged particle trace points defined with the **TRPOIN (p. 2021)** command.

Menu Paths

Main Menu>General Postproc>Plot Results>Dele Trace Pt

TRPLIS, *NTRP1*, *NTRP2*, *TRPINC*, *Opt*

Lists charged particle trace points.

POST1 (p. 51): Trace Points (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NTRP1*, *NTRP2*, *TRPINC

List points from *NTRP1* to *NTRP2* (defaults to *NTRP1*) in steps of *TRPINC* (defaults to 1). If *NTRP1* = ALL, *NTRP2* and *TRPINC* are ignored and all trace points are listed. If *NTRP1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

Opt

Opt = LOC lists the trace point number location (X, Y, Z). Default.

Opt = PART lists the trace point number particle settings (velocity, charge, mass).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Lists the charged particle trace points in the active display coordinate system (**DSYS (p. 582)**). Trace points are defined with the **TRPOIN (p. 2021)** command.

Menu Paths

Main Menu>General Postproc>Plot Results>List Trace Pt

TRPOIN, *X*, *Y*, *Z*, *VX*, *VY*, *VZ*, *CHRG*, *MASS*

Defines a point through which a charged particle trace will travel.

POST1 (p. 51): Trace Points (p. 56)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X, Y, Z

Coordinate location of the trace point (in the active coordinate system). If *X* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

VX, VY, VZ

Particle velocities in the X, Y and Z directions (in the active coordinate system).

CHRG

Particle charge.

MASS

Particle mass.

Notes

Defines a point through which a charged particle trace (**PLTRAC (p. 1442)**) will travel. Multiple points (50 maximum) may be defined which will result in multiple particle traces. Use **TRPLIS (p. 2020)** to list the currently defined trace points and **TRPDEL (p. 2020)** to delete trace points.

The VX, VY, VZ, CHRG, and MASS arguments only apply to charged particles.

Menu Paths

Main Menu>General Postproc>Plot Results>Defi Trace Pt

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TRTIME, *TIME*, *SPACING*, *OFFSET*, *SIZE*, *LENGTH*

Defines the options used for the PLTRAC (p. 1442) (charged particle trace) command.

POST1 (p. 51): Animation (p. 53)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TIME

Total Trace Time (seconds) (defaults to 0, which is the full particle trace).

SPACING

Particle spacing in seconds (defaults to 0).

OFFSET

Particle offset in seconds (defaults to 0). It is used internally in the **ANFLOW (p. 139)** macro to produce an animation of charged particle motion in an electric or magnetic field.

SIZE

Particle size (defaults to 0, which is a line).

LENGTH

Particle length fraction (defaults to .1).

Command Default

Full charged particle trace.

Notes

The **TRTIME (p. 2022)** command varies the type of **PLTRAC (p. 1442)** display produced. Charged particle traces follow a particle's path in the forward and backward direction of travel. The DOF selected determines the color of the particle trace. *SPACING* defines the particle spacing in seconds from adjacent particles in the stream line. *OFFSET* defines the offset in seconds from the spacing set by the *SPACING* argument.

LENGTH defines the particle length fraction. The default value (.1), means the particle occupies 10% of the trace region, and the other 90% is a color-coded line.

SIZE sets the radius of the particle. Use *SPACING*, *OFFSET* and *LENGTH* only when *SIZE* is nonzero (that is, the particle is bigger than the line).

Menu Paths

Main Menu>General Postproc>Plot Results>Time Interval

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TSHAP,*Shape*

Defines simple 2D and 3D geometric surfaces for target segment elements.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Shape

Specifies the geometric shapes for target segment elements TARGE169 and TARGE170.

LINE

Straight line (2D, 3D) (Default for 2D)

PARA

Parabola (2D, 3D)

ARC

Clockwise arc (2D)

CARC

Counterclockwise arc (2D)

CIRC

Complete circle (2D)

TRIA

Three-node triangle (3D) (Default for 3D)

TRI6

Six-node triangle (3D)

QUAD

Four-node quadrilateral (3D)

QUA8

Eight-node quadrilateral (3D)

CYLI

Cylinder (3D)

CONE

Cone (3D)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SPHE

Sphere (3D)

PILO

Pilot node (2D, 3D)

POINT

Point (rigid surface node) (2D, 3D)

Notes

Use this command to specify the target segment shapes for the rigid target surface associated with surface-to-surface contact (TARGE169, CONTA172 (2D) and TARGE170, CONTA174 (3D)), 3D beam-to-beam contact (TARGE170 and CONTA177), and 3D line-to-surface contact (TARGE170 and CONTA177). Once you issue **TSHAP (p. 2023)**, all subsequent target elements generated via the direct element generation technique will have the same shape, until you issue **TSHAP (p. 2023)** again with a different *Shape* value.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

---

/**TSPEC**, *TCOLOR*, *TSIZE*, *TXTHIC*, *PANGLE*, *IANGLE*

Creates annotation text attributes (GUI).

GRAPHICS (p. 16): Annotation (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TCOLOR

Text color (0 ≤ *TCOLOR* ≤ 15):

0

Black.

1

Red-Magenta.

2

Magenta.

3

Blue-Magenta.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4

Blue.

5

Cyan-Blue.

6

Cyan.

7

Green-Cyan.

8

Green.

9

Yellow-Green.

10

Yellow.

11

Orange.

12

Red.

13

Dark Gray.

14

Light Gray.

15

White.

TSIZE

Text size factor.

TXTHIC

Text thickness key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

normal.

2

twice as thick.

3

three times as thick.

4

four times as thick.

PANGLE

Text path angle (0.0 < *angle* < 360.0).

IANGLE

Text italic angle (0.0 < *angle* < 45.0).

Notes

This command defines annotation text attributes to control certain characteristics of the text created via **/TLABEL (p. 2007)**. This command is generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if annotation is used.

The command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Annotation>Create Annotation

TSRES,*Array*

Defines an array of key times at which the time-stepping strategy changes.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Array

Identifies an *N*x1x1 array parameter containing the key times at which the heat transfer time-stepping strategy changes (the time step is reset to the initial time step based on **DELTIM (p. 515)** or **NSUBST (p. 1287)** settings). The array name must be enclosed by % signs (for example, %*array*%). See ***DIM (p. 530)** for more information on array parameters.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Time values in the array parameter must be in ascending order and must not exceed the time at the end of the load step as defined on the **TIME (p. 2000)** command. The time increment between time points in the array list must be larger than the initial time step defined on the **DELTIM (p. 515)** or **NSUBST (p. 1287)** command. Time values must also fall between the beginning and ending time values of the load step. For multiple load step problems, you must either change the parameter values to fall between the beginning and ending time values of the load step or reissue the command with a new array parameter. To clear the array parameter specification, issue **TSRES (p. 2026)**,ERASE. Results can be output at the requested time points if the array or time values in the array are also specified in the **OUTRES (p. 1336)** command using *FREQ*=%`array`%. Use this command to reset the time-stepping strategy within a load step. You may need to reset the time-stepping strategy when using tabular time-varying boundary conditions.

See Steady-State Thermal Analysis of the *Thermal Analysis Guide* for more information on applying boundary conditions via tabular input. See Transient Thermal Analysis of the *Thermal Analysis Guide* for more information on defining the key time array.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Preprocessor>Loads>Load Step Opts>Time/Frequenc>Time and Substeps

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time - Time Step

Main Menu>Solution>Load Step Opts>Time/Frequenc>Time and Substeps

---

TUNIF, *TEMP*

Assigns a uniform temperature to all nodes.

`SOLUTION` (p. 38): FE Body Loads (p. 48)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

TEMP

Uniform temperature assigned to the nodes. If a *TEMP* value is not specified, the uniform temperature is set to zero.

Command Default

Set the uniform temperature to the reference temperature (defined via the **TREF (p. 2013)** command).

Notes

TUNIF (p. 2027) is a convenient form of the more general **BFUNIF (p. 259)** command.

In a transient or nonlinear thermal analysis, the uniform temperature is used during the first iteration of a solution as follows:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- as the starting nodal temperature, except where temperatures are explicitly specified (**D (p. 483)**, **DK (p. 541)**)
- to evaluate temperature-dependent material properties.

In a structural analysis, the uniform temperature is used as the *default* temperature for thermal strain calculations and material property evaluation, except where body load temperatures are specified (**BF (p. 221)**, **BFE (p. 237)**, **BFK (p. 248)**, **LDREAD (p. 1036)**). In other scalar field analyses, the uniform temperature is used for material property evaluation.

Because **TUNIF (p. 2027)** (or **BFUNIF (p. 259)**,TEMP) is step-applied in the first iteration, issue a **BF (p. 221)**,ALL,TEMP,*Value* command to ramp on a uniform temperature load.

The command default sets the uniform temperature to the reference temperature defined via the **TREF (p. 2013)** command only (and *not* the **MP (p. 1160)**,REFT command).

If using the command default to set the uniform temperature (to the reference temperature set via **TREF (p. 2013)**), you can convert temperature-dependent secant coefficients of thermal expansion (SCTEs) from the definition temperature to the uniform temperature. To do so, issue the **MPAMOD (p. 1166)** command.

In a mode-superposition harmonic or transient analysis, you must apply the load in the modal portion of the analysis. Mechanical APDL calculates a load vector and writes it to the MODE file, which you can apply via the **LVSCALE (p. 1094)** command.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Define Loads>Apply>Electric>Boundary>Temperature>Uniform Temp

Main Menu>Preprocessor>Loads>Define Loads>Apply>Magnetic>Boundary>Temperature>Uniform Temp

Main Menu>Preprocessor>Loads>Define Loads>Apply>Structural>Temperature>Uniform Temp

Main Menu>Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>Uniform Temp

Main Menu>Preprocessor>Loads>Define Loads>Settings>Uniform Temp

Main Menu>Solution>Define Loads>Apply>Electric>Boundary>Temperature>Uniform Temp

Main Menu>Solution>Define Loads>Apply>Magnetic>Boundary>Temperature>Uniform Temp

Main Menu>Solution>Define Loads>Apply>Structural>Temperature>Uniform Temp

Main Menu>Solution>Define Loads>Apply>Thermal>Temperature>Uniform Temp

Main Menu>Solution>Define Loads>Settings>Uniform Temp

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TVAR, *KEY*

Changes time to the cumulative iteration number.

POST26 (p. 58): Controls (p. 59)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KEY

Time key:

0

Time is used for the variable *TIME*.

1

NCUMIT is used for the variable *TIME*.

Command Default

TIME is the variable *TIME*.

Notes

Changes the meaning of the time variable to the cumulative iteration number (NCUMIT) variable. Data can be read from the file, printed, and displayed as a function of NCUMIT rather than time. All POST26 descriptions applying to TIME then apply to NCUMIT.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Data

/TXTRE, *Lab*, *NUM*, *N1*, *N2*, *NINC*

Controls application of texture to selected items.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Lab

You can apply texture according to the following labels:

ELEM

Apply texture to elements *N1* through *N2* in steps of *NINC*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AREA

Apply texture to areas *N1* through *N2* in steps of *NINC*.

VOLU

Apply texture to volumes *N1* through *N2* in steps of *NINC*.

CM

Apply texture to the component named in *N1*. *N2* and *NINC* are ignored.

ON, OFF

Sets the specified texture display on or off. All other fields are ignored.

File

If Lab = File, the command format is **/TXTRE (p. 2029)**, File, *Key_Index*, *Fname*, *Fext*, --, *Format* (This variant of the command is applicable to 2D drivers).

Key_Index

The texture index associated with the file. If the number fifty-one (51) is used, the imported bitmap will be used as the window's logo.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Fext

Filename extension (eight-character maximum).

--

Unused field.

Format

The file format. If *Format* = 0, the file is a pixmap (Linux) or Bitmap (PC). The file cannot contain a compressed image, and the PC file must be 8 or 24 bit BI_RGB format. If *Format* = 1 or JPEG, then the file is in JPEG (Joint Photographic Experts Group) format. If *Format* = 2 or PNG, then the file is in PNG (Portable Network Graphics) format.

NUM

Select the texture index number from the following list:

0

No Texturing

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Aluminum

2

Aluminum, Brushed

3

Steel With Bumps

4

Steel, Embossed

5

Iron

6

Steel, Pattern

7

Steel, Riveted

8

Steel, Scratched

9

Tin

10

Metal

11

Steel, Etched

12

Metal, Hot

13

Iron, Grainy

14

Metal, Rusty

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15

Brick

16

Block

17

Wood

18

Wood, Light

19

Wood, Walnut

20

Plastic, Hard Blue

21

Plastic, Light Blue

22

Plastic, Hard Red

31

Gold

32

Brass

33

Silver

34

Plastic, Black

35

Plastic, Ivory

36

Plastic, Blue

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37

Plastic, Red

38

Plastic, Yellow

39

Plastic, Green

40

Plastic, Brown

N1*, *N2*, *NINC

Apply texture to *Lab* items numbered *N1* through *N2* in steps of *NINC* (defaults to 1). If *Lab* = CM, then N1 is used to for the component name and *N2* and *NINC* are ignored. If *Lab* = ELEM, AREA, or VOLU and *N1* = blank or ALL, then the specified texture will be applied to all entities of type *Lab*. If *N1* = P, then graphical picking is enabled.

Command Default

No texture (**/TXTRE (p. 2029)**,DEFA)

Notes

This command is available for 3D Open GL devices. 2D devices are supported **only** for the *Lab* = *File* variation of the command, allowing imported bitmaps to be used for texturing and annotation. Textures can affect the speed of many of your display operations. You can increase the speed by temporarily turning the textures off (**Utility Menu> PlotCtrls> Style> Texturing(3D)> Display Texturing**). This menu selection toggles your textures on and off. When textures are toggled off, all of the texture information is retained and reapplied when texturing is toggled back on.

For some displays, the texture will appear distorted because of a technique used to enhance 3D displays (**/DV3D (p. 584)**,TRIS,1). Disabling this function (**/DV3D (p. 584)**,TRIS,0) will improve the quality of some texture displays. Disabling the TRIS option of the **/DV3D (p. 584)** command will slow down 3D displays significantly. Be sure to reapply the TRIS option after you obtain a satisfactory output.

Specifying **/TXTRE (p. 2029)**,DEFA removes all texturing.

Menu Paths

Utility Menu>PlotCtrls>Style>Texturing(3D)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/TYPE, *WN*, *Type*

Defines the type of display.

GRAPHICS (p. 16): Style (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

Type

Display type. Defaults to ZBUF for raster mode displays or BASIC for vector mode displays:

BASIC or 0

Basic display (no hidden or section operations).

SECT or 1

Section display (plane view). Use the **/CPLANE (p. 426)** command to define the cutting plane.

HIDC or 2

Centroid hidden display (based on item centroid sort).

HIDD or 3

Face hidden display (based on face centroid sort).

HIDP or 4

Precise hidden display (like HIDD but with more precise checking). Because all facets are sorted, this mode can be extremely slow, especially for large models.

CAP or 5

Capped hidden display (same as combined SECT and HIDD with model in front of section plane removed).

ZBUF or 6

Z-buffered display (like HIDD but using software Z-buffering).

ZCAP or 7

Capped Z-buffered display (same as combined SECT and ZBUF with model in front of section plane removed).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ZQSL or 8

QSLICE Z-buffered display (same as SECT but the edge lines of the remaining 3D model are shown).

HQSL or 9

QSLICE precise hidden display (like ZQSL but using precise hidden).

Command Default

ZBUF for raster mode displays; BASIC for vector mode displays.

Notes

Defines the type of display, such as section display or hidden-line display. Use the **/DEVICE (p. 523)** command to specify either raster or vector mode.

The SECT, CAP, ZCAP, ZQSL, and HQSL options produce section displays. The section or "cutting" plane is specified on the **/CPLANE (p. 426)** command as either normal to the viewing vector at the focus point (default), or as the working plane.

When you use PowerGraphics, the section display options (Section, Slice, and Capped) use different averaging techniques for the interior and exterior results. Because of the different averaging schemes, anomalies may appear at the transition areas. In many cases, the automatically computed MIN and MAX values will differ from the full range of interior values. You can lessen the effect of these anomalies by issuing **AVRES (p. 212)**,,FULL (**Main Menu> General Post Proc> Options for Outp**). This command sets your legend's automatic contour interval range according to the minimum and maximum results found throughout the entire model.

With PowerGraphics active (**/GRAPHICS (p. 866)**,POWER), the averaging scheme for surface data with interior element data included (**AVRES (p. 212)**,,FULL) and multiple facets per edge (**/EFACET (p. 602)**,2 or **/EFACET (p. 602)**,4) will yield differing minimum and maximum contour values depending on the Z-Buffering options (**/TYPE (p. 2034)**,,6 or **/TYPE (p. 2034)**,,7). When the Section data is not included in the averaging schemes (**/TYPE (p. 2034)**,,7), the resulting absolute value for the midside node is significantly smaller.

The HIDC, HIDD, HIDP, ZBUF, ZQSL, and HQSL options produce displays with "hidden" lines removed. Hidden lines are lines obscured from view by another element, area, etc. The choice of non-Z-buffered hidden-line procedure types is available only for raster mode (**/DEVICE (p. 523)**) displays. For vector mode displays, all non-Z-buffered "hidden-line" options use the same procedure (which is slightly different from the raster procedures). Both geometry and postprocessing displays may be of the hidden-line type. Interior stress contour lines within solid elements can also be removed as hidden lines, leaving only the stress contour lines and element outlines on the visible surfaces. Midside nodes of elements are ignored on postprocessing displays. Overlapping elements will not be displayed.

The ZBUF, ZCAP, and ZQSL options use a specific hidden-line technique called software Z-buffering. This technique allows a more accurate display of overlapping surfaces (common when using Boolean operations or **/ESHAPE (p. 665)** on element displays), and allows smooth shaded displays on all interactive graphics displays. Z-buffered displays can be performed faster than HIDP and CAP type displays for large models. See also the **/LIGHT (p. 1049)**, **/SHADE (p. 1783)**, and **/GFILE (p. 858)** commands for additional options when Z-buffering is used.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Hidden-Line Options

TYPE, *ITYPE*

Sets the element type attribute pointer.

PREP7 (p. 22): Meshing (p. 29)
PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITYPE

Assign a type number to the elements (defaults to 1).

Command Default

ITYPE = 1.

Notes

Assigns an element-type number to subsequently defined elements. The number refers to the element-type number (*ITYPE*) defined with via **ET (p. 686)**. You can display type numbers (**/PNUM (p. 1457)**).

In some cases, the program can proceed with a meshing operation even when no logical element type has been assigned via **TYPE (p. 2036)** or ***X*ATT**,,,*TYPE*. For more information, see the discussion for setting element attributes in Meshing Your Solid Model.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Default Attribs

Main Menu>Preprocessor>Modeling>Create>Elements>Elem Attributes

Main Menu>Preprocessor>Modeling>Operate>Extrude>Elem Ext Opts

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

U Commands

/UCMD, *Cmd*, *SRNUM*

Assigns a user-defined command name.

APDL (p. 19): Abbreviations (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Cmd

User-defined command name. Only the first four characters are significant. Must not conflict with any Mechanical APDL command name or any user unknown-command macro name.

SRNUM

User subroutine number (1 to 10) programmed for this command. For example, the command **/UCMD (p. 2037)**,MYCMD,3 will execute subroutine USER03 whenever the command **MYCMD** is entered. Use a blank command name to disassociate *SRNUM* from its command. For example, **/UCMD (p. 2037)**,,3 removes **MYCMD** as a command.

Notes

Assigns a user-defined command name to a user-programmable (system-dependent) subroutine. This feature allows user-defined commands to be programmed into Mechanical APDL. Once programmed, this command can be input to the program like other commands, and can also be included in the Mechanical APDL start-up file.

Up to 10 subroutines are available for user-defined commands (USER01 to USER10). You must have system permission, system access, and knowledge to write, compile, and link the appropriate subprocessors into Mechanical APDL at your site.

All routines should be written in FORTRAN. For more information about FORTRAN compilers, refer to either the Ansys, Inc. Windows Installation Guide or the Ansys, Inc. Linux Installation Guide for details specific to your platform or operating system.

The USER01 routine is commented and should be listed from the distribution media (system dependent) for more details.

Issue **/UCMD (p. 2037)**,STAT to list all user-defined command names.

Because a user-programmed command is a nonstandard use of the program, the verification of any Mechanical APDL run incorporating these commands is your responsibility. In any contact with Mech-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

anical APDL customer support regarding the performance of a custom version of Mechanical APDL, explicitly state that a user-programmable feature has been used.

See User-Programmable Features (UPFs) in the *Advanced Analysis Guide* for a general description of user-programmable features and the Guide to User-Programmable Features in the *Programmer's Reference* for a detailed description of these features.

See ***ULIB (p. 2051)** for another way of defining user commands.

This command is valid only at the Begin Level.

Menu Paths

This command cannot be accessed from a menu.

/UDOC,*WIND*,*Class*,*Key*,

Determines position and content for the multi-legend options.

GRAPHICS (p. 16): Labeling (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WIND

The window number to which the command applies. (defaults to 1)

Class

The type (and relative importance) of legend item being displayed:

CNTR

Contour legend. This legend item is controlled separately from the other legend items (see note below).

DATE

The items in the DATE class include the date and time, or the Mechanical APDL graphical logo (**/PLOPTS (p. 1435)**,LOGO,1). This item is shown by default in all plots.

GWIN

The items in the GWIN class include the entity acronyms that appear in the legend of a multiplot of entities (Nodes, Elements, Keypoints, Lines, Areas, Volumes). GWIN items are shown by default for all **GPLOT (p. 866)** displays.

TYPE

Items in the TYPE class include the plot type (e.g. ELEMENTS, MATERIALS, NODAL SOLUTIONS, etc.). TYPE items are shown by default in all plots.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TYP2

Items in the TYP2 class include supplementary type information, such as DMAX and SMAX for nodal solutions. TYP2 items are shown by default in all plots.

INUM

Items in the INUM class include the number labels generated by the **/PNUM (p. 1457)** command. This class is displayed by default in all plots that contain **/PNUM (p. 1457)** information.

BCDC

The items in the BCDC class include labels created by the **/PBC (p. 1356)** command. This class is shown by default in all plots which contain **/PBC (p. 1356)** information.

VECT

Items in the VECT class include labels created by the **PLVECT (p. 1445)** command. This class is shown by default for all **PLVECT (p. 1445)** plots.

SURF

The items in the SURF class include labels from the **/PSF (p. 1537)** legend. This class is shown by default on all plots of surface boundary conditions.

BODY

Items from the BODY class include labels from the **/PBF (p. 1360)** legend. This class is shown by default in all plots of body forces.

PSTA

Items from the PSTA class include stress scaling statistics, such as the **/SSCALE (p. 1847)** setting. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

VIEW

The items in the VIEW class include view statistics. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

MISC

The items in the MISC class include supplementary labels like /EXPANDED and Stress Section Cross Section. This class is not shown as the default for any type of plot, and must be specifically referenced to display the included data.

KEY

Switch:

OFF or 0 --	Do not display the legend.
LEFT, RIGHT, TOP or BOTTOM --	If the value for *Class* is CNTR, these are the four acceptable values for the contour legend position.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LEFT or RIGHT -- If any value other than CNTR is used for *Class*, these are the two acceptable values for the text data.

Notes

The legend classes conform to the controls specified in the window options panel (**PlotCtrls> Window Controls> Window Options**). In many instances, the legend controls specified with the **/PLOPTS (p. 1435)** command will take precedence and override **/UDOC (p. 2038)** specifications. For instance:

/PLOPTS (p. 1435),LEG1,OFF will disable the TYPE, TYP2, INUM, and MISC classes, regardless of the **/UDOC (p. 2038)** settings.

/PLOPTS (p. 1435),LEG2,OFF will disable the VIEW class, regardless of the **/UDOC (p. 2038)** settings.

/PLOPTS (p. 1435),LEG3,OFF will disable the PSTA class, regardless of the **/UDOC (p. 2038)** settings.

All items in a class are listed with the same X coordinate (except for contours). The contents of the text classes are dumped onto the display window from top to bottom, in order of class importance.

The font specification for text items that are included in the user-specified legends are controlled with the **/DEVICE (p. 523)** command (**PlotCtrls> Font Controls> Anno/Graph Font**).

The floating point values for the data presented in the legend(s) are controlled by the **/GFORMAT (p. 859)** command.

Menu Paths

Utility Menu>PlotCtrls>Style>MultiLegend Options>Contour Legend

Utility Menu>PlotCtrls>Style>MultiLegend Options>Text Legend

/UI,*Func*,*Type*,*Format*,*Screen*,*Color*,*Krev*,*Orient*,*Compress*,*Quality*

Activates specified GUI dialog boxes.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Func

Label identifying the dialog box to be activated:

HELP

Activates the online help system.

VIEW

Activates the Pan, Zoom, Rotate dialog box

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WPSE

Activates the Working Plane Settings dialog box.

WPVI

Activates the Offset Working Plane dialog box.

RESULT

Activates the Query Picking Menu for reviewing results.

QUERY

Activates the Query Picked Entities (preprocess) dialog box.

COPY

Activates the Hard Copy dialog box.

ANNO

Activates the 2D Annotation dialog box.

AN3D

Activates the 3D Annotation dialog box.

SELECT

Activates the Select Entities dialog box.

NSEL

Activates a picking menu to select nodes.

ESEL

Activates a picking menu to select elements.

KSEL

Activates a picking menu to select keypoints.

LSEL

Activates a picking menu to select lines.

ASEL

Activates a picking menu to select areas.

VSEL

Activates a picking menu to select volumes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

REFRESH

Refreshes the graphics window (non-UI mode only).

COLL

Controls the collapse of the Mechanical APDL main menu when a **FINISH (p. 758)** command is issued. See *Type* below for a discussion of the arguments.

Type

Label identifying the type of select operation. Valid only for the following *Func* labels; NSEL, ESEL, KSEL, LSEL, ASEL, and VSEL:

S

Select a new set.

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Label identifying the type of results data to be queried. Valid only for *Func* = RESULT:

NODE

Nodal solution data (h-elements only).

ELEMENT

Element solution data.

Label specifying the behavior of the Mechanical APDL main menu after a **FINISH (p. 758)** command is issued. User interaction with the main menu is unaffected. Valid only for *Func* = COLL:

YES, 1 or blank

Allows the Main Menu to collapse after **FINISH (p. 758)** command.

NO or 0

Prevents Main Menu collapse after **FINISH (p. 758)** command.

If *Func* = **COPY**, and *Type* = **SAVE,** command format is **/UI (p. 2040)**,*Func*,*Type*,*Format*,*Screen*,*Color*,*Krev*,*Orient*,*Compress*,*Quality*. The remaining fields (after *Type*) identify the portion of the screen, the type of file, and the method of display that the portion of the screen is to be saved as.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Format

TIFF

Tagged Image File Format.

BMP

(PC only) Bitmap (Windows) file format.

WMF

(PC only) Windows Metafile format.

EMF

(PC only) Enhanced Metafile format.

JPEG

JPEG (Joint Photographic Experts Group) file format.

Screen

FULL

Saves the entire screen in the specified format.

GRAPH

Saves only the Mechanical APDL graphic window.

Color

MONO

A two color (black and white) file is saved.

GRAY

The specified file format is saved in gray scale.

COLOR

The file is saved at the specified color depth.

Krev

NORM

Saves file as shown on the screen.

REVERSE

Saves file with the background color reversed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Orient

LANDSCAPE

Saves file in landscape mode.

PORTRAIT

Saves file in portrait mode.

Compress

YES

Compresses TIFF files and EPS files with TIFF preview (default).

NO

Saves files with no compression.

Quality

1,2,,,100

JPEG quality index, with 100 being the maximum quality level.

Notes

Allows you to activate specified GUI dialog boxes directly in either GUI or non-GUI mode.

The **/UI (p. 2040)** command itself is valid in any processor, however certain dialog boxes are accessible only in a particular processor (for example, **/UI (p. 2040)**,RESULT,... is valid only in the General Postprocessor).

Mechanical APDL JPEG software is based in part on the work of the Independent JPEG Group, Copyright 1998, Thomas G. Lane.

Menu Paths

This command cannot be accessed from a menu.

UIMP,*MAT*,*Lab1*,*Lab2*,*Lab3*,*VAL1*,*VAL2*,*VAL3*

Defines constant material properties (GUI).

PREP7 (p. 22): Materials (p. 24)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT

Material number.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab1, Lab2, Lab3

Material property labels (see **MP (p. 1160)** for valid labels).

VAL1, VAL2, VAL3

Values corresponding to three labels.

Notes

UIMP (p. 2044) is generated by the Graphical User Interface (GUI) and appears in the log file (`Jobname.log`) if material properties are specified via the **Material Properties** dialog. The command is *not* intended to be typed in directly during an analysis (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Change Mat Props>Material Models

Main Menu>Preprocessor>Material Props>Material Models

Main Menu>Solution>Load Step Opts>Other>Change Mat Props>Material Models

---

/**UIS**, *Label*, *VALUE*

Controls the GUI behavior.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Behavior control key:

BORD

Controls the functionality of the mouse buttons for dynamic viewing mode only. When *Label* = BORD, the three values that follow control the functionality of the LEFT, MIDDLE and RIGHT buttons, respectively (see below).

MSGPOP

Controls which messages from the Mechanical APDL error-message subroutine are displayed in a message dialog box.

REPLOT

Controls whether or not an automatic replot occurs after functions affecting the model are executed.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ABORT

Controls whether or not the program displays dialog boxes to show the status of an operation in progress and to cancel that operation.

DYNA

Controls whether the dynamic mode preview is a bounding box or the edge outline of the model. This label only applies to 2D display devices (that is, **/SHOW (p. 1785)**,XII or **/SHOW (p. 1785)**,WIN32). This "model edge outline" mode is only supported in PowerGraphics (**/GRAPHICS (p. 866)**,POWER) and is intended for element, line, results, area, or volume displays.

PICK

Controls how graphical entities are highlighted from within the the **Select** menu.

POWER

Controls whether or not PowerGraphics is active when the GUI is initiated. Mechanical APDL default status is PowerGraphics "ON"; this command is used (placed in the `start.ans` file) when full graphics is desired on start up.

DPRO

Controls whether or not the input window displays a dynamic prompt. The dynamic prompt shows the correct command syntax for the command, as you are entering it.

UNDO

Controls whether or not the session editor includes nonessential commands or comments in the file it creates. You can use this option to include comments and other materials in the session editor file.

LEGE

Controls whether or not the multi-legend is activated when you start the GUI. The multi-legend enables you to specify the location of your legend items in each of the five graphics windows. You can place this option in your `start.ans` file and have the GUI start with the legend items in a pre-specified location.

PBAK

Controls whether or not the background shading is activated when you start the GUI. You can place this option in your `start.ans` file.

ZPIC

Controls the sorting order for entities that are coincident (directly in front of or behind each other) to a picked spot on your model. When you pick a spot on your model that could indicate two or more entities, a message warns you of this condition, and a list of the coincident entities can be generated. The *VALUE* term (below) will determine the sort order.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HPOP

Controls the prioritization of your GUI windows when the contents are ported to a plot or print file (**/UI (p. 2040)**,COPY,SAVE). OpenGL (3D) graphics devices require that the Mechanical APDL graphics window contents be set in front of all overlying windows in order to port them to a printer or a file. This operation can sometimes conflict with **/NOERASE (p. 1249)** settings. See the *VALUE* term (below) to determine the available control options.

VALUE

Values controlling behavior if *Label* = BORD:

(These values control the operation according to syntax : **/UIS (p. 2045)**,BORD,*LEFT*,*MIDDLE*,*RIGHT*)

1

PAN, controls dynamic translations.

2

ZOOM, controls zoom, and dynamic rotation about the view vector.

3

ROTATE, controls dynamic rotation about the screen X and Y axes.

Note:

You can designate any value for any button, or designate the same value for all three buttons. If no value is specified, default is LEFT = PAN, MIDDLE = ZOOM and RIGHT = ROTATE.

Values controlling behavior if *Label* = MSGPOP:

0

All messages displayed.

1

Only notes, warnings, and errors displayed.

2

Only warnings and errors displayed (default).

3

Only errors displayed.

Values controlling behavior if *Label* = REPLOT:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

No automatic replot.

1

Automatic replot (default).

Values controlling behavior if *Label* = ABORT:

ON

Display status and cancellation dialog boxes (default).

OFF

Do not display status and cancellation dialog boxes.

1

Same as ON.

0

Same as OFF.

Values controlling behavior if *Label* = DYNA:

0

Use model edge outline when possible (default).

1

Use bounding box preview.

Values controlling behavior if *Label* = PICK:

0

Picked keypoints and nodes are enclosed by a square. Picked lines are overlaid by a thicker line. Picked areas, volumes, and elements (non-point/non-line) are redrawn with highlighting colors. However, if the pick is a box, circle, or polygon pick, the highlighting for all entitles consists only of a square placed around the entity's centroid.

1

Picked entities are not highlighted.

2

5.1 highlighting (that is, no XOR).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3

Picked entities are highlighted as in *VALUE* = 0, except that, for a box, circle, or polygon pick, the picked areas, volumes, and elements (non-point/non-line) are redrawn with highlighting colors. This technique is slower than the *VALUE* = 0 technique.

Values controlling behavior if *Label* = POWER:

0

Start GUI in Full Graphics mode.

1

Start GUI in PowerGraphics mode (default).

Values controlling behavior if *Label* = DPRO:

0 or OFF

Do not display the dynamic prompt.

1 or ON

Display the dynamic prompt (default).

Values controlling behavior if *Label* = UNDO:

0 or None

Do not suppress any commands (default).

1 or Comment

Write the nonessential commands to the session editor file as comments (with a ! at the beginning).

2 or Remove

Do not write nonessential commands or comments.

Values controlling behavior if *Label* = LEGE:

0 or OFF

Start GUI with the enhanced legend off (default).

1 or ON

Start GUI with the enhanced legend capability activated.

Values controlling behavior if *Label* = PBAK:

0 or OFF

Start the GUI with the no background shading (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 or ON

Start the GUI with background shading activated.

Values controlling behavior if *Label* = HPOP:

0 or OFF

No rewrite operations are performed to compensate for items that obscure or overlay the graphics window (default).

1 or ON

The Graphics screen contents are replotted to ensure that they are situated in front of all other windows. If **/NOERASE (p. 1249)** is detected, this operation is suppressed.

Notes

Controls certain features of the Graphical User Interface (GUI), including whether Mechanical APDL displays dialog boxes to show you the status of an operation (such as meshing or solution) in progress and to enable you to cancel that operation. Issue **/UIS (p. 2045)**,STAT for current status. Issue **/UIS (p. 2045)**,DEFA to reset default values for all labels. Issue **/UIS (p. 2045)**,*Label*,STAT and **/UIS (p. 2045)**,*Label*,DEFA for status and to reset a specific *Label* item.

A **/UIS (p. 2045)**,HPOP,1 command employs a fast redraw method which does not allow entering the legend logic for a **/PLOPTS (p. 1435)**,INFO,1 or **/PLOPTS (p. 1435)**,INFO,2 command. However, the legend *is* redrawn for **/PLOPTS (p. 1435)**,INFO,3 because that command also allows a fast redraw.

This command is valid in any processor.

Menu Paths

Utility Menu>MenuCtrls>Message Controls

Utility Menu>PlotCtrls>Device Options

/UPF, *RoutineName*

Links a user-programmable routine into Mechanical APDL.

SESSION (p. 11): Files (p. 12)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

RoutineName

Name of a user-programmable routine (`filename.ext`) to link.

Notes

For more information, see Using the /UPF Command for Linux Systems and Using the /UPF Command for Windows Systems, both found in the *Guide to User-Programmable Features*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

---

*ULIB,*Fname*,*Ext*,--

Identifies a macro library file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Command Default

No macro library file.

Notes

Identifies a macro library file for the ***USE (p. 2059)** command.

A library of macros allows blocks of often-used commands to be stacked and executed from a single file. The macro blocks must be enclosed within block identifier and terminator lines as shown in this example:

Example 19: Macro Blocks

```
ABC! Any valid alphanumeric name (32 characters maximum)
!  identifying this data block
---! Mechanical APDL data-input commands
---
---
/EOF! Terminator for this data block
XYZ! Identify another data block (if desired)
---! Mechanical APDL data-input commands
---
---
/EOF! Terminator for this data block
(etc.)
```

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To add comment lines to a macro block, place them anywhere *within* the macro block, including directly on the lines where the macro block identifier and the macro block terminator appear as shown in the example. Do not place comment lines (or any other lines) outside of a macro block.

The name of the macro library file is identified for reading via ***ULIB (p. 2051)**. The name of the macro block is identified via ***USE (p. 2059)**.

Commands within the macro block are copied to a temporary file (of the macro block name) during the ***USE (p. 2059)** operation and executed as if a macro file of that name had been created. The temporary file is deleted after it has been used.

Macro block names should be acceptable file names (system-dependent) and should not match user-created macro file names, as the user-created macro file is used first (if it exists) before the library file is searched.

Macro blocks may be stacked in any order. Branching (***GO (p. 864)** or ***IF (p. 927)**) external to the macro block is not allowed.

This command is valid in any processor.

Menu Paths

Utility Menu>Macro>Execute Data Block

UNDELETE,*Option*,*Nstart*,*Nend*

Removes results sets from the group of sets selected for editing.

AUX3 (p. 61): Results Files (p. 61)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Option

Specifies which sets are to be removed from the selected sets.

SET

Specifies one or more particular sets in the results file that are to be removed from the group of sets selected for deletion.

ALL

Removes all selected sets that are currently selected for deletion.

Nstart

The first set to be removed from the set selected for deletion.

Nend

The final set to be removed from the set selected for deletion. This field is used only if operating on more than one sequential set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Use this command if you have previously marked a set for deletion (with the **DELETE (p. 513)** command) and now wish to keep that set instead of deleting it.

The **UNDELETE (p. 2052)** command is valid only in the results file editing processor (auxiliary processor AUX3), and, like the other AUX3 commands, it only affects the data steps index (DSI), time (TIM), loadstep, substep and cumulative step iteration (LSP) records in the results file.

Menu Paths

This command cannot be accessed from a menu.

UNDO, *Keyword*

Enables the user to modify or save commands issued since the last RESUME or SAVE.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Keyword

NEW

Create an editable GUI window that allows the user to alter the commands issued since the most recent **SAVE (p. 1645)** or **RESUME (p. 1601)** operations (GUI only).

Notes

This command invokes the session editor, a text window displaying all program operations since the last **SAVE (p. 1645)** or **RESUME (p. 1601)**.

You can modify command parameters, delete whole sections of text, and even save a portion of the command string to a separate file. The file is named `jobname000.cmds`, with each subsequent save operation incrementing the filename by one.

The session editor file can be changed only by the session editor. If you rename your database file outside of Mechanical APDL and then resume that database, the session editor displays the old filename.

For more information, see Using the Session Editor.

Menu Paths

Main Menu>Preprocessor>Session Editor

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/UNITS, *Label*, *LENFACT*, *MASSFACT*, *TIMEFACT*, *TEMPFACT*, *TOFFST*, *CHARGEFACT*, *FORCEFACT*, *HEATFACT*

Annotates the database with the system of units used.

DATABASE (p. 13): Set Up (p. 13)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Label

Label to denote the system of units used in this job:

USER

User-defined system (default).

SI

International system (m, kg, s, K).

MKS

MKS system (m, kg, s, °C).

uMKS

μMKS system (μm, kg, s, °C).

CGS

CGS system (cm, g, s, °C).

MPA

MPA system (mm, Mg, s, °C).

BFT

U. S. Customary system using feet (ft, slug, s, °F).

BIN

U. S. Customary system using inches (in, lbf*s^2/in, s, °F).

If *Label* = USER, the remaining fields on this command may be used to enter conversion factors that are appropriate for the user-defined system of units.

LENFACT

Conversion factor to meter (m). Default = 1.

MASSFACT

Conversion factor to kilogram (kg). Default = 1.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TIMEFACT

Conversion factor to second (s). Default = 1.

TEMPFACT

Conversion factor to kelvin (K). Default = 1.

TOFFST

Temperature offset from absolute zero in kelvin. Default = 0.

CHARGEFACT

Conversion factor to coulomb. Default = 1.

FORCEFACT

Conversion factor to newton (N). Default = 1.

HEATFACT

Conversion factor to joule (J). Default = 1.

Command Default

User-defined units.

Notes

Allows the user to set a marker in the database indicating the system of units used. The setting may be reviewed with the **/STATUS (p. 1865)** command at the Begin level. The units label and conversion factors on this command are for user convenience only and have no effect on the analysis or data. That is, **/UNITS (p. 2054)** *will not* convert database items from one system to another (for example, from U. S. Customary to SI, etc.). The units setting will be written to the file of IGES data [**IGESOUT (p. 932)** or **CDWRITE (p. 293)**], which can then be read by many programs that read IGES files. The user must still use consistent units for the results to be valid.

If you choose the MKS system of units, the EPZRO option for the **EMUNIT (p. 632)** command is set to 8.85 e-12 F/m. (EPZRO specifies alternate free-space permittivity.)

For micro-electromechanical systems (MEMS), where dimensions are on the order of microns, see the conversion factors in System of Units in the *Coupled-Field Analysis Guide*.

If you use the Mechanical APDL ADAMS Interface to export model information to the ADAMS program, the **/UNITS (p. 2054)** command is required to ensure the correct transfer of data between Mechanical APDL and ADAMS. You can choose a predefined unit system label (*Label* = SI, CGS, etc.) or you can select the user-defined system option (*Label* = USER) and input the appropriate conversion factors (*LENFACT*, *MASSFACT*, *TIMEFACT*, and *FORCEFACT*). The conversion factors are written to the ADAMS input file `Jobname.MNF` to correctly generate the load. For more information, see Export to ADAMS.

All differences between the base solution units used by the Mechanical APDL and CFX solvers are noted in the Mechanical APDL output file. Unit conversions are applied automatically to all loads transferred

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

unless *Label* = USER. Unit conversions are not applied to any of the loads transferred between the Mechanical APDL and CFX solvers if they use a user-defined unit system.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

UNPAUSE

Restores use of a temporarily released product license.

SESSION (p. 11): Run Controls (p. 11)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

The **UNPAUSE (p. 2056)** command restores use of a temporarily released (paused) product license. The command is valid only *after* a previously issued **PAUSE (p. 1356)** command.

When use of the product license is paused via the **PAUSE (p. 1356)** command, no other operation (other than **SAVE (p. 1645)** or **/EXIT (p. 708)**) is possible until you issue the **UNPAUSE (p. 2056)** command.

For more information, see the documentation for the **PAUSE (p. 1356)** command and the Ansys Licensing Guide.

Menu Paths

This command cannot be accessed from a menu.

UPCOORD, *FACTOR*, *Key*

Modifies the coordinates of the active set of nodes, based on the current displacements.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FACTOR

Scale factor for displacements being added to nodal coordinates. If *FACTOR* = 1.0, the full displacement value will be added to each node, 0.5, half the displacement value will be added, etc. If *FACTOR* = -1, the full displacement value will be subtracted from each node, etc.

Key

Key for zeroing displacements in the database:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OFF

Do not zero the displacements (default).

ON

Zero the displacements.

Notes

The **UPCOORD (p. 2056)** command uses displacements stored in the Mechanical APDL database, and not those contained within the results file, `Jobname.rst`. Nodal coordinates are updated each time the command is issued. After updating, both the nodal displacements and rotations are set to zero if *Key* = ON.

For structural solutions with an updated mesh, unless the coefficient matrix is otherwise reformed (for example, a new analysis or **NLGEOM (p. 1231)**,ON) it should first be reformed by issuing a **KUSE (p. 1002)**,-1 command.

UPCOORD (p. 2056) should not be issued between load steps in structural analysis.

For a multiphysics simulation where a CFD or electromagnetic field is being coupled to a structure undergoing large displacements, all (or a portion) of the surrounding field mesh may take part in the structural solution to "move" with the displacing structure. You can use the **UPCOORD (p. 2056)** command with a suitable *FACTOR* to update the coordinates of the nodes using the newly computed displacements. The mesh will now conform with the displaced structure for subsequent field solutions. However, the mesh should always be restored to its original location by using an **UPCOORD (p. 2056)**,*FACTOR* command before performing any subsequent structural solutions. This is true for both repeated linear solutions, and for nonlinear restarts. (All saved displacements are relative to the original mesh location.)

Caution:

Orientation nodes for beams and pipes always have zero displacements. Therefore, although this command may alter the locations of other beam and pipe nodes, it has no effect on orientation nodes. Carefully inspect the element coordinate systems on the updated model.

This command is not intended to replace either the large-displacement or birth-and-death capability.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>Updt Node Coord

Main Menu>Solution>Load Step Opts>Other>Updt Node Coord

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UPGEOM, *FACTOR*, *LSTEP*, *SBSTEP*, *Fname*, *Ext*, *--*, *UPESYS*

Adds displacements from a previous analysis and updates the geometry to the deformed configuration.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

FACTOR

Multiplier for displacements being added to coordinates. The value 1.0 adds the full value of the displacements to the geometry of the finite element model. Defaults to 1.0.

LSTEP

Load step number of data to be imported. Defaults to the last load step.

SBSTEP

Substep number of data to be imported. Defaults to the last substep.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The field must be input (no default).

Ext

Filename extension (eight-character maximum).

The extension must be .RST.

--

Unused field.

UPESYS

Update behavior for the element coordinate system (**ESYS (p. 685)**):

0 -- Do not update the element coordinate system (default).

1 -- Update the element coordinate system to match the material orientation from a previous analysis.

Notes

This command updates the geometry of the finite element model according to the displacement results of the previous analysis and creates a revised geometry at the deformed configuration. This command works on all nodes (default) or on a selected set of nodes. If this command is issued repeatedly, it creates a revised geometry of the finite element model in a cumulative fashion, that is, it adds displacement

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

results on the previously generated deformed geometry. The solid model geometry is not updated by this command.

When **UPGEOM (p. 2058)** is issued, the current finite element model is overwritten by finite element information from the results file. For this reason, it is important that the finite element information in the results file matches the finite element model in which the nodal coordinates are being updated. No changes should be made to the model before the **UPGEOM (p. 2058)** command is issued.

UPESYS = 1 is available for homogeneous structural solid elements (SOLID185, SOLID186, and SOLID187) only and generates only Cartesian coordinate systems. The option is especially useful when conducting a loop test when orthotropic material is used. For more information, see Nonlinear Static Analysis with Inverse Solving in the *Structural Analysis Guide*.

Caution:

Orientation nodes for beams and pipes always have zero displacements. Therefore, although this command may alter the locations of other beam and pipe nodes, it has no effect on orientation nodes. Carefully inspect the element coordinate systems on the updated model.

Menu Paths

Main Menu>Preprocessor>Modeling>Update Geom

---

*USE,*Name*,*ARG1*,*ARG2*,*ARG3*,*ARG4*,*ARG5*,*ARG6*,*ARG7*,*ARG8*,*ARG9*,*AR10*,*AR11*,*AR12*,*AR13*,*AR14*,*AG15*,*AR16*,*AR17*,*AR18*

Executes a macro file.

APDL (p. 19): Macro Files (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Name

Name (32 characters maximum, beginning with a letter) identifying the macro file or a macro block on a macro library file.

ARG1, ARG2, ARG3, . . . , AR18

Values passed into the file or block where the parameters ARG1 through ARG9 and AR10 through AR18 are referenced. Values may be numbers, alphanumeric character strings (up to 32 characters enclosed in single quotes), parameters (numeric or character) or parametric expressions. See below for additional details.

Notes

Causes execution of a macro file called *Name*, or, if not found, a macro block "*Name*" on the macro library file (***ULIB (p. 2051)**). Argument values (numeric or character) are passed into the file or block and substi-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tuted for local parameters ARG1, ARG2, ..., AR18. The file *Name* may also be executed as an "unknown command" (that is, without the ***USE (p. 2059)** command name) as described below.

A macro is a sequence of Mechanical APDL commands (as many as needed) recorded in a file or in a macro block in a library file (specified with the ***ULIB (p. 2051)** command). The file or block is typically executed via ***USE (p. 2059)**. In addition to command, numerical and alphanumeric data, the macro can include parameters which will be assigned numerical or alphanumerical character values when the macro is used. Use of the macro can be repeated (within a do-loop, for example) with the parameters incremented.

A macro is defined within a run by enclosing a sequence of data input commands between ***CREATE (p. 434)** and ***END (p. 635)** commands. The data input commands are passive (not executed) while being written to the macro file. The macro file (without ***CREATE (p. 434)** and ***END (p. 635)**) can also be created external to Mechanical APDL.

Up to 99 specially named scalar parameters, ARG1 to AR99, are *locally* available to each macro:

- The prefix for the first nine parameters is ARG, and the prefix for the remaining 90 parameters is AR.
- A local parameter is not affected by, nor does it affect, other parameters, even those of the same name, which are used outside of the macro. The only way a local parameter can affect, or be affected by, parameters outside the macro is if values are passed out of, or into, the macro by an argument list.
- Parameters ARG1 through AR18 can have their values (numeric or character) passed via the argument list on ***USE (p. 2059)**. (ARG1 through AR19 can be passed as arguments on the unknown-command macro.) Parameters AR19 through AR99 (AR20 through AR99 in the unknown-command macro) are available solely for use within the macro; they cannot be passed via an argument list.
- Local parameters are available to do-loops and to **/INPUT (p. 948)** files processed within the macro. In addition to an ARG1--AR99 set for each macro, another ARG1--AR99 set is available external to all macros, local to "non-macro" space.

A macro is exited after its last line is executed. Macros may be nested (such as a ***USE (p. 2059)** or an unknown command within a macro). Each nested macro has its own set of 99 local parameters. Only one set of local parameters can be active at a time and that is the set corresponding to the macro currently being executed or to the set external to all macros (if any). When a nested macro completes execution, the previous set of local parameters once again becomes available. Issue ***STATUS (p. 1863)**,ARGX to view current macro parameter values.

An alternate way of executing a macro file is via the unknown-command route. If a command unknown to Mechanical APDL is entered, a search for a file of that name (plus a `.mac` suffix) is made. If the file exists, it is executed, if not, the "unknown command" message is output. Thus, you can write your own commands in terms of other Mechanical APDL commands. The procedure is similar to issuing ***USE (p. 2059)** with the unknown command in the *Name* field. For example, the command **CMD**,10,20,30 is internally similar to ***USE (p. 2059)**,CMD,10,20,30. The macro file named `cmd.mac` is executed with the three parameters. The ***USE (p. 2059)** macro description also applies to the unknown-command macro, except that various directories are searched and a suffix (`.mac`) is assumed. Also, a macro library file is not searched.

A three-level directory search for the unknown-command macro file may be available. The search order may be: 1) a high-level system directory, 2) the log-in directory, and 3) the local (working) directory. Issue

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/PSEARCH (p. 1535) to change the directory search path. For an unknown command **CMD**, the first file named `cmd.mac` found to exist in the search order is executed. The command can be input in lower-, upper-, or mixed-case; however, it converts to uppercase automatically before the file name search occurs. On systems that preserve the case as it was input, a file matching the upper-case name is used first, followed by a file with the matching the lower-case name, and finally a file matching the mixed-case name. All macro files placed in the `apdl` directory must be upper-case.

Because undocumented commands exist in Mechanical APDL, you should issue the command intended for the macro file name to ensure that the unknown-command message is output in the processor where it is to be used. If the macro is to be used in other processors, the other processors must also be checked.

This command is valid in any processor.

Menu Paths

Utility Menu>Macro>Execute Data Block

/USER, *WN*

Conveniently resets /FOCUS (p. 771) and /DIST (p. 536) to USER.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

Notes

Conveniently resets scale parameters to USER on the **/FOCUS (p. 771)** and **/DIST (p. 536)** commands. Scale parameters will be internally respecified to those used for the last display. Convenient when the last scale parameters were automatically calculated. User specified parameters hold until changed or removed (**/AUTO (p. 207)**). Parameters may be reset on the individual commands after this command has been issued.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>View Settings>Automatic Fit Mode

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

USRCAL, *Rnam1*, *Rnam2*, *Rnam3*, *Rnam4*, *Rnam5*, *Rnam6*, *Rnam7*, *Rnam8*, *Rnam9*

Allows user-solution subroutines to be activated or deactivated.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Rnam1, Rnam2, Rnam3, . . . , Rnam9

User-defined solution subroutine names to be activated. Up to nine may be defined on one command or multiple commands may be used. If *Rnam1* = ALL, activate all valid user subroutines. If *Rnam1* = NONE, deactivate all valid user subroutines. All characters are required:

USREFL

Allows user defined scalar field (body force) loads.

USERCV

Allows user defined convection (surface) loads.

USERPR

Allows user defined pressure (surface) loads.

USERFX

Allows user-defined heat flux (surface) loads.

USERCH

Allows user-defined charge density (surface) loads.

USERFD

Computes the complex load vector for the frequency domain logic.

USEROU

Allows user supplied element output.

USOLBEG

Allows user access before each solution.

ULDBEG

Allows user access before each load step.

USSBEG

Allows user access before each substep.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UITBEG

Allows user access before each equilibrium iteration.

UITFIN

Allows user access after each equilibrium iteration.

USSFIN

Allows user access after each substep.

ULDFIN

Allows user access after each load step.

USOLFIN

Allows user access after each solution.

UANFIN

Allows user access at end of run.

UELMATX

Allows user access to element matrices and load vectors.

UTIMEINC

Allows a user-defined time step, overriding the program-determined time step.

UCNVRG

Allows user-defined convergence checking, overriding the program-determined convergence.

Command Default

No user-solution subroutines are active (even if linked into the program).

Notes

Allows certain user-solution subroutines to be activated or deactivated (system-dependent). This command only affects the subroutines named. Other user subroutines (such as user elements, user creep, etc.) have their own activation controls described with the feature.

The UAnBeg subroutine that allows user access at the start of a run does not require activation by this command; it is automatically activated when the program is started.

The routines are commented and should be listed after performing a custom installation from the distribution media for more details. See also the *Advanced Analysis Guide* for a general description of user-programmable features.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You must have system permission, system access, and knowledge to write, compile, and link the appropriate subroutines into the program at your site.

All routines should be written in FORTRAN. (For more information about FORTRAN compilers, refer to either the Ansys, Inc. Windows Installation Guide or the Ansys, Inc. Linux Installation Guide for details specific to your platform or operating system.)

Issue **USRCAL (p. 2062)**,STAT to list the status of these user subroutines.

Because a user-programmed subroutine is a nonstandard use of the program, the verification of any Mechanical APDL run incorporating these commands is entirely your responsibility. In any contact with customer support regarding the performance of a custom version of Mechanical APDL, explicitly state that a user-programmable feature has been used.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Other>User Routines

Main Menu>Solution>Load Step Opts>Other>User Routines

USRDOF,*Action*,*DOF1*,*DOF2*,*DOF3*,*DOF4*,*DOF5*,*DOF6*,*DOF7*,*DOF8*,*DOF9*,*DOF10*

Specifies the degrees of freedom for the user-defined element USER300.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

One of the following command operations:

DEFINE

Specify the degrees of freedom (DOFs). This value is the default.

LIST

List all previously specified DOFs.

DELETE

Delete all previously specified DOFs.

DOF1*, *DOF2*, *DOF3*, . . . , *DOF10

The list of DOFs.

Notes

The **USRDOF (p. 2064)** command specifies the degrees of freedom for the user-defined element USER300.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Although you can intersperse other commands as necessary for your analysis, issue the **USRDOF (p. 2064)** command as part of the following *general sequence* of commands:

1. Issue the **ET (p. 686)** command for element USER300, followed by the related **TYPE (p. 2036)** command.
2. Issue both the **USRELEM (p. 2065)** and **USRDOF (p. 2064)** commands (in either order).
3. Define your element using USER300.

The DOF list (*DOF1* through *DOF10*) can consist of up to 10 DOFs. Use any valid and appropriate DOF (such as UX, UY, UZ, ROTX, ROTY, ROTZ, AX, AY, AZ, VX, VY, VZ, PRES, WARP, TEMP, VOLT, MAG, EMF, and CURR).

You can specify a maximum of 10 DOFs per **USRDOF (p. 2064)** command. To define additional DOFs, issue the command again.

The maximum number of DOFs for a user-defined element--the number of nodes times the number of DOFs per node--cannot exceed 480.

To learn more about user-defined elements, see Creating a New Element in the *Programmer's Reference*.

Menu Paths

This command cannot be accessed from a menu.

USRELEM,*NNODES*,*NDIM*,*KeyShape*,*NREAL*,*NSAVEVARS*,*NRSLTVAR*,*KEYANSMAT*,*NINTPNTS*,*KESTRESS*,*KEYSYM*

Specifies the characteristics of the user-defined element USER300.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NNODES

The number of nodes.

NDIM

The number of dimensions (of nodal coordinates). Valid values are 2 or 3.

KeyShape

One of the following element shape options:

ANYSHAPE

Any shape (that is, no specified shape). This value is the default.

POINT

Point.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LINE

Straight line.

TRIAN

Triangle.

QUAD

Quadrilateral. This shape can be degenerated to a triangle.

TET

Tetrahedron.

BRICK

Brick. This shape can be degenerated to a wedge, pyramid, or tetrahedron.

NREAL

The number of real constants.

NSAVEVARS

The number of saved variables.

NRSLTVAR

The number of variables saved in results files.

KEYANSMAT

Key for element formulation control:

0

Create your own material codes within the element formulation. In this case, the real constants are available to input material properties. You can also input linear material properties via **MP (p. 1160)** and **MPDATA (p. 1168)** commands .

1

Use standard material routines or the `UserMat` subroutine to form structural material data. Material properties must be input in the standard way (as you would for non-user-defined elements). This value is invalid when *KeyShape* = ANYSHAPE.

NINTPNTS

The maximum number of integration points (used when *KEYANSMAT* = 1).

KESTRESS

Key for the element stress state (used when *KEYANSMAT* = 1):

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Plane stress elements.

1

Axisymmetric elements.

2

Plane strain elements.

3

3D solid elements.

4

3D solid-shell elements.

5

Generalized plane strain elements.

6

Beam elements.

7

Link/truss elements.

8

3D shell elements.

9

Axisymmetric shell elements.

KEYSYM

Key for specifying whether element stiffness matrices are symmetric or unsymmetric:

0

Symmetric.

1

Unsymmetric.

Notes

The **USRELEM (p. 2065)** command specifies the characteristics of the user-defined element USER300.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Although you can intersperse other commands as necessary for your analysis, issue the **USRELEM (p. 2065)** command as part of the following *general sequence* of commands:

1. Issue the **ET (p. 686)** command for element USER300, followed by the related **TYPE (p. 2036)** command.
2. Issue both the **USRELEM (p. 2065)** and **USRDOF (p. 2064)** commands (in either order).
3. Define your element using USER300.

The number of real constants (*NREAL*) can refer to geometry quantities, material quantities, or any parameters for element formulation.

ANSYS saves variables in the `.esav` file to preserve element data when you specify a positive *NSAVEVARS* value. When *KEYANSMAT* = 0, all variables of both material and kinematic formulation are saved. When *KEYANSMAT* = 1, only the variables for kinematic formulation (such as deformation gradient tensor) are saved; in this case, the material routine saves all necessary material data automatically.

Element data saved in results files (*NRSLTVAR*) are accessible only as nonsummable miscellaneous data. ANSYS saves stress and total strain data for structural elements in the `.rst` file automatically (as it does for equivalent variables such as thermal gradient and thermal flux in thermal elements); therefore, *NRSLTVAR* does not need to include stress and total strain data.

To learn more about creating user-defined elements, see Creating a New Element in the *Programmer's Reference*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

V Commands

V, *P1*, *P2*, *P3*, *P4*, *P5*, *P6*, *P7*, *P8*

Defines a volume through keypoints.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

P1

Keypoint defining starting corner of volume. If *P1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

P2

Keypoint defining second corner of volume.

P3

Keypoint defining third corner of volume.

P4

Keypoint defining fourth corner of volume.

P5

Keypoint defining fifth corner of volume.

P6

Keypoint defining sixth corner of volume.

P7

Keypoint defining seventh corner of volume.

P8

Keypoint defining eighth corner of volume.

Notes

Defines a volume (and its corresponding lines and areas) through eight (or fewer) existing keypoints. Keypoints must be input in a continuous order. The order of the keypoints should be around the bottom and then the top. Missing lines are generated "straight" in the active coordinate system and assigned

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the lowest available numbers (**NUMSTR (p. 1301)**). Missing areas are generated and assigned the lowest available numbers.

Solid modeling in a toroidal coordinate system is not recommended.

Certain faces may be condensed to a line or point by repeating keypoints. For example, use **V (p. 2069)**,*P1*,*P2*,*P3*,*P3*,*P5*,*P6*,*P7*,*P7* for a triangular prism or **V (p. 2069)**,*P1*,*P2*,*P3*,*P3*,*P5*,*P5*,*P5*,*P5* for a tetrahedron.

Using keypoints to produce partial sections in **CSYS (p. 441)** = 2 can generate anomalies; check the resulting volumes carefully.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volumes>Arbitrary>Through KPs

---

V2DOPT, *GEOM*, *NDIV*, *HIDOPT*, *NZONE*

Specifies 2D/axisymmetric view factor calculation options.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

GEOM

Choice of geometry:

0

Planar (default).

1

Axisymmetric

NDIV

Number of divisions for axisymmetric geometry (that is, the number of circumferential segments). Default is 50. There is no maximum limit if *HIDOPT* = 0; the maximum is 90 if *HIDOPT* = 1. If *NDIV* is $\leq$ 6, it is reset to 50.

For more information, see View Factors of Axisymmetric Bodies in the *Mechanical APDL Theory Reference*.

HIDOPT

Viewing option:

0

Hidden (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Non-hidden

NZONE

Number of zones (that is, the number of rays emanating from a surface) for view factor calculation. This is used if *HIDOPT* = 0.

If *NZONE* is blank, it is set to 0 and the view factor algorithm sets the number of rays. (See Command Default below.)

If *NZONE* is < 0 or > 1000, it is set to 200.

Command Default

By default, a planar geometry is assumed (*GEOM* = 0) and the hidden viewing option is used (*HIDOPT* = 0).

The view factor algorithm sets the number of rays as follows, depending on whether a planar or axisymmetric geometry is specified:

- For *GEOM* = 0, *HIDOPT* = 0 and *NZONE* = 0, the number of zones used in view the factor calculation is 200.
- For *GEOM* = 1, *HIDOPT* = 0, and *NZONE* = 0, the number of zones used in the view factor calculation is 20.

Notes

V2DOPT (p. 2070) sets 2D view factor calculation options for the radiosity solver method. For 2D view factor calculations, the ray-emanation method is used.

The geometry type can be either 2D planar (default) or axisymmetric. For the axisymmetric case, you can define the number of circumferential segments (defaults to 20). You can also specify the hidden or non-hidden viewing option (defaults to hidden) and the number of zones for the view factor calculation. For more information, see Process for Using the Radiosity Solver Method in the *Thermal Analysis Guide* .

Menu Paths

Main Menu>Preprocessor>Radiation Opts>View Factor

Main Menu>Radiation Opt>Radiosity Meth>View Factor

Main Menu>Solution>Radiation Opts>View Factor

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VA,*A1*,*A2*,*A3*,*A4*,*A5*,*A6*,*A7*,*A8*,*A9*,*A10*

Generates a volume bounded by existing areas.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

A1, A2, A3, . . . , A10

List of areas defining volume. The minimum number of areas is 4. If *A1* = ALL, use all selected (**ASEL (p. 192)**) areas and ignore *A2* to *A10*. If *A1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *A1*.

Notes

This command conveniently allows generating volumes from regions having more than eight keypoints (which is not allowed with the **V (p. 2069)** command). Areas may be input in any order. The exterior surface of a **VA (p. 2072)** volume must be continuous, but holes may pass completely through it.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Volume by Areas

Main Menu>Preprocessor>Modeling>Create>Volumes>Arbitrary>By Areas

***VABS**,*KABSR*,*KABS1*,*KABS2*,*KABS3*

Applies the absolute value function to array parameters.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

KABSR

Absolute value of results parameter:

0 --

Do not take absolute value of results parameter (ParR).

1 --

Take absolute value.

KABS1

Absolute value of first parameter:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0 --

Do not take absolute value of first parameter (Par1 or ParI).

1 --

Take absolute value.

KABS2

Absolute value of second parameter:

0 --

Do not take absolute value of second parameter (Par2 or ParJ).

1 --

Take absolute value.

KABS3

Absolute value of third parameter:

0 --

Do not take absolute value of third parameter (Par3 or ParK).

1 --

Take absolute value.

Command Default

Do not use absolute values.

Notes

Applies an absolute value to parameters used in certain ***V*XX** and ***M*XX** operations. Typical absolute value applications are of the form:

ParR = |f(|Par1|)|

or

ParR = |(|Par1| o |Par2|)|

The absolute values are applied to each input parameter value before the operation and to the result value after the operation. Absolute values are applied before the scale factors so that negative scale factors may be used. The absolute value settings are *reset* to the default (no absolute value) after each ***V*XX** or ***M*XX** operation. Use ***VSTAT (p. 2167)** to list settings.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

VADD, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Adds separate volumes to create a single volume.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1, NV2, NV3, . . . , NV9

Numbers of volumes to be added. If *NV1* = ALL, add all selected volumes and ignore *NV2* to *NV9*. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Adds separate volumes to create a single volume. The original volumes (and their corresponding areas, lines and keypoints) will be deleted by default (**BOPTN (p. 269)**). See the **BOPTN (p. 269)** command for the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. Concatenated entities are not valid with this command.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Add>Volumes

Main Menu>Preprocessor>Modeling>Operate>Booleans>Add>Volumes

VARDEL, *NVAR*

Deletes a variable (GUI).

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NVAR

The reference number of the variable to be deleted. *NVAR* is as defined by **NSOL (p. 1281)**, **ESOL (p. 672)**, etc.

Notes

Deletes a POST26 solution results variable. This command is generated by the Graphical User Interface (GUI). It appears in the log file (`Jobname.log`) if a POST26 variable is deleted from the **Defined Time-History Variables** dialog box.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The command is *not* intended to be typed in directly in a Mechanical APDL session (although it can be included in an input file for batch input or for use with **/INPUT (p. 948)**).

Menu Paths

Main Menu>TimeHist Postpro>Define Variables

Main Menu>TimeHist Postpro>Elec&Mag>Circuit>Define Variables

VARNAM, *IR*, *Name*

Names (or renames) a variable.

POST26 (p. 58): Set Up (p. 58)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IR

Reference number of the variable (2 to NV (**NUMVAR (p. 1302)**)).

Name

Thirty-two character name for identifying variable on printouts and displays. Embedded blanks are compressed for output.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

Main Menu>TimeHist Postpro>Settings>List

VATT, *MAT*, *REAL*, *TYPE*, *ESYS*, *SECNUM*

Associates element attributes with the selected, unmeshed volumes.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

MAT*, *REAL*, *TYPE*, *ESYS*, *SECNUM

Material number, real constant set number, type number, coordinate system number, and section number to be associated with selected, unmeshed volumes.

Notes

These element attributes will be used when the volumes are meshed. If a volume does not have attributes associated with it (by this command) at the time it is meshed, the attributes are obtained from the then current **MAT (p. 1111)**, **REAL (p. 1579)**, **TYPE (p. 2036)**, **ESYS (p. 685)**, and **SECNUM (p. 1685)** command settings. Reissue the **VATT (p. 2075)** command (before volumes are meshed) to change the attributes. A zero (or blank) argument removes the corresponding association.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If any of the arguments *MAT*, *REAL*, *TYPE*, *ESYS* or *SECNUM* are defined as -1, then that value will be left unchanged in the selected set.

In some cases, the program can proceed with a volume meshing operation even when no logical element type has been assigned via **VATT (p. 2075)**,,,*TYPE* or **TYPE (p. 2036)**. For more information, see the discussion on setting element attributes in Meshing Your Solid Model of the *Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>All Volumes

Main Menu>Preprocessor>Meshing>Mesh Attributes>Picked Volumes

---

VCLEAR, *NV1*, *NV2*, *NINC*

Deletes nodes and volume elements associated with selected volumes.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1*, *NV2*, *NINC

Delete mesh for volumes *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and mesh for all selected volumes (**VSEL (p. 2163)**) is deleted. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

Deletes *all* nodes and volume elements associated with selected volumes (regardless of whether the nodes or elements are selected). Nodes shared by adjacent meshed volumes and nodes associated with non-volume elements will not be deleted. Attributes assigned as a result of **VATT (p. 2075)** are maintained. In the program's response to the command, if a volume, area, line, or keypoint is tallied as "cleared," it means either its node or element reference was deleted.

Menu Paths

Main Menu>Preprocessor>Meshing>Clear>Volumes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VCOL, *NCOL1*, *NCOL2*

Specifies the number of columns in matrix operations.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

NCOL1

Number of columns to be used for Par1 with ***MXX*** operations. Defaults to whatever is needed to fill the result array.

NCOL2

Number of columns to be used for Par2 with ***MXX*** operations. Defaults to whatever is needed to fill the result array.

Command Default

Fill all locations of the result array from the specified starting location.

Notes

Specifies the number of columns to be used in array parameter matrix operations. The size of the sub-matrix used is determined from the upper left starting array element (defined on the operation command) to the lower right array element (defined by the number of columns on this command and the number of rows on the ***VLEN (p. 2136)** command).

The default *NCOL* is calculated from the maximum number of columns of the result array (the ***DIM (p. 530)** column dimension) minus the starting location + 1. For example, ***DIM (p. 530)**,R,,1,10 and a starting location of R(1,7) gives a default of 4 columns (starting with R(1,7), R(1,8), R(1,9), and R(1,10)). Repeat operations automatically terminate at the last column of the result array. Existing values in the rows and columns of the results matrix remain unchanged where not overwritten by the requested input or operation values.

The column control settings are *reset* to the defaults after each ***MXX*** operation. Use ***VSTAT (p. 2167)** to list settings.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/VCONE, *WN*, *PHI*

Defines the view cone angle for perspective displays.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

PHI

View cone angle (0.0 to 85.°) to define perspective. Use *PHI* = 45.0° for typical perspective. Increase angle for more perspective, decrease angle for less. If the distance (**/DIST (p. 536)**) is not specified, it will be automatically calculated to give full window magnification. If the distance is also specified, *PHI* controls both the perspective and the magnification. The larger the angle, the more the perspective and the less the magnification. Defaults to 0.0 (no perspective).

Command Default

0.0 degrees (no perspective (parallel projection)).

Notes

Perspective shows the true depth of the object in the display. A variable magnification results since the back plane of the object is further from the observer than the front plane. The largest magnification occurs at the front plane. With perspective, the magnification factor (MAGF) is not only a function of the distance from the object, but also the window shape and the perspective (or view cone) angle Φ as follows:

$$MAGF = \frac{\ell / 2}{(d)\ TAN\phi}$$

where ℓ, for square windows, is the largest in-plane vertical or horizontal dimension, d is the distance from the observer to the plane of ℓ (usually the front plane of the object), and Φ is the view cone angle (defined with the **/VCONE (p. 2078)** command). The bigger the cone angle, the more the perspective. The magnification factor proportionally decreases with increasing Φ. The distance can be defined with the **/DIST (p. 536)** or the **/FOCUS (p. 771)** command. Note, the distance input on the **/DIST (p. 536)** command is equal to d only if the focus point is located on the plane of ℓ. It is recommended that if a general perspective is desired (that is, not any specific cone angle), use Φ = 45.0 (since TAN(45.0) = 1.0) and let the d value be automatically calculated for full window magnification.

Note that any number of **/DIST (p. 536)**, **/FOCUS (p. 771)**, and **/VCONE (p. 2078)** combinations can be used to produce the same magnification. Distances less than the object depth will produce views from within the object.

A magnification factor of 1.0 just fills the window. If the automatic scaling option is used (**/AUTO (p. 207)**), the magnification factor is fixed at 0.91 (to allow a 10% margin around the object) and d is automatically calculated for the given **/VCONE (p. 2078)** and **/FOCUS (p. 771)** values. Any value of Φ between 0.0 and 85.0 (usually 45.0) may be used to activate the perspective. Views from inside the object are not possible

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when d is automatically calculated (use manual scaling (**/USER (p. 2061)**) along with **/DIST (p. 536)** specification).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>View Settings>Perspective View

VCROSS, *LabXR, LabYR, LabZR, LabX1, LabY1, LabZ1, LabX2, LabY2, LabZ2*

Forms element table items from the cross product of two vectors.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabXR, LabYR, LabZR

Label assigned to X, Y, and Z-component of resultant vector.

LabX1, LabY1, LabZ1

X, Y, and Z-component of first vector label.

LabX2, LabY2, LabZ2

X, Y, and Z-component of second vector label.

Notes

Forms labeled result items for the selected element from the cross product of two vectors:

{*LabXR*, *LabYR*, *LabZR*} = {*LabX1*, *LabY1*, *LabZ1*} X {*LabX2*, *LabY2*, *LabZ2*}

Data must be in a consistent coordinate system. Labels are those associated with the **ETABLE (p. 687)** command.

Menu Paths

Main Menu>General Postproc>Element Table>Cross Product

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VCUM, *KEY*

Allows array parameter results to add to existing results.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

KEY

Accumulation key:

0 --

Overwrite results.

1 --

Add results to the current value of the results parameter.

Command Default

Overwrite results.

Notes

Allows results from certain ***VXX*** and ***MXX*** operations to overwrite or add to existing results. The cumulative operation is of the form:

ParR = ParR + ParR(Previous)

The cumulative setting is *reset* to the default (overwrite) after each ***VXX*** or ***MXX*** operation. Use ***VSTAT (p. 2167)** to list settings.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VDDAM, *VF*, *VA*, *VB*, *VC*

Specifies the velocity spectrum computation constants for the analysis of shock resistance of shipboard structures.

SOLUTION (p. 38): Spectrum Options (p. 41)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VF

Direction-dependent velocity coefficient for elastic or elastic-plastic analysis option (Default = 0).

VA, VB, VC

Coefficients for the DDAM velocity spectrum equations. See Dynamic Design Analysis Method in the *Mechanical APDL Theory Reference*. Default for these coefficients is zero.

Notes

This command specifies velocity coefficients to analyze shock resistance of shipboard equipment. These coefficients are used to compute mode coefficients according to the equations given in Dynamic Design Analysis Method in the *Mechanical APDL Theory Reference*. The form of these equations is based on the Naval NRL Dynamic Design Analysis Method. This command, along with the **ADDAM (p. 96)** and **SED (p. 1704)** commands, is used with the spectrum (**ANTYPE (p. 162)**,SPECTR) analysis as a special purpose alternative to the **SV (p. 1886)**, **FREQ (p. 776)**, and **SVTYP (p. 1887)** commands.

In order to perform a DDAM spectrum analysis using a units system other than BIN (default), you must specify the units system complying with the mass and length units of the model using the **/UNITS (p. 2054)** command. Issue the **/UNITS (p. 2054)** command before defining the shock spectrum computation constants (**VDDAM (p. 2081)**). The **VDDAM (p. 2081)** command is not supported with the user-defined units system (*Label* = USER on the **/UNITS (p. 2054)** command).

DDASPEC (p. 503) may alternatively be used to calculate spectrum coefficients.

This command is also valid in PREP7.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Spectrum>DDAM Options

Main Menu>Solution>Load Step Opts>Spectrum>DDAM Options

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VDELE, *NV1*, *NV2*, *NINC*, *KSWP*

Deletes unmeshed volumes.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1*, *NV2*, *NINC

Delete volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and all selected volumes (**VSEL (p. 2163)**) are deleted. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

KSWP

Specifies whether keypoints, lines, and areas are also deleted:

0

Delete volumes only (default).

1

Delete volumes, as well as keypoints, lines, and areas attached to the specified volumes but not shared by other volumes.

Menu Paths

Main Menu>Preprocessor>Modeling>Delete>Volume and Below

Main Menu>Preprocessor>Modeling>Delete>Volumes Only

VDGL, *NV1*, *NV2*, *NINC*

Lists keypoints of a volume that lie on a parametric degeneracy.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1*, *NV2*, *NINC

List keypoints that lie on a parametric degeneracy on volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* will be ignored and keypoints on all selected volumes (**VSEL (p. 2163)**) will be listed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). You may also substitute a component name for *NV1* (ignore *NV2* and *NINC*).

Notes

See the *Modeling and Meshing Guide* for details about parametric degeneracies.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>List Degen Volus

Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>List Degen Volus

---

VDOT, *LabR*, *LabX1*, *LabY1*, *LabZ1*, *LabX2*, *LabY2*, *LabZ2*

Forms an element table item from the dot product of two vectors.

POST1 (p. 51): Element Table (p. 52)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LabR

Label assigned to dot product result.

LabX1, LabY1, LabZ1

X, Y, and Z-component of first vector label.

LabX2, LabY2, LabZ2

X, Y, and Z-component of second vector label.

Notes

Forms labeled result items for the selected element from the dot product of two vectors:

$$LabR = \{LabX1, LabY1, LabZ1\} \cdot \{LabX2, LabY2, LabZ2\}$$

Data must be in a consistent coordinate system. Labels are those associated with the **ETABLE (p. 687)** command.

Menu Paths

Main Menu>General Postproc>Element Table>Dot Product

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VDRAG, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *NLP1*, *NLP2*, *NLP3*, *NLP4*, *NLP5*, *NLP6*

Generates volumes by dragging an area pattern along a path.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA6

List of areas in the pattern to be dragged (6 maximum if using keyboard entry). If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NA1* = ALL, all selected areas will be swept along the path. A component name may also be substituted for *NA1*.

NLP1, NLP2, NLP3, . . . , NLP6

List of lines defining the path along which the pattern is to be dragged (6 maximum if using keyboard entry). Must be a continuous set of lines. To be continuous, adjacent lines must share the connecting keypoint (the end keypoint of one line must also be first keypoint of the next line).

Notes

Generates volumes (and their corresponding keypoints, lines, and areas) by sweeping a given area pattern along a characteristic drag path. If the drag path consists of multiple lines, the drag direction is determined by the sequence in which the path lines are input (*NLP1*, *NLP2*, etc.). If the drag path is a single line (*NLP1*), the drag direction is from the keypoint on the drag line that is closest to the first keypoint of the given area pattern to the other end of the drag line.

The magnitude of the vector between the keypoints of the given pattern and the first path keypoint remains constant for all generated keypoint patterns and the path keypoints. The direction of the vector relative to the path slope also remains constant so that patterns may be swept around curves. Lines are generated with the same shapes as the given pattern and the path lines.

Keypoint, line, area, and volume numbers are automatically assigned (beginning with the lowest available values (**NUMSTR (p. 1301)**)). Adjacent lines use a common keypoint, adjacent areas use a common line, and adjacent volumes use a common area. For best results, the entities to be dragged should be orthogonal to the start of the drag path. Drag operations that produce an error message may create some of the desired entities prior to terminating.

If element attributes have been associated with the input area via the **AATT (p. 85)** command, the opposite area generated by the **VDRAG (p. 2084)** operation will also have those attributes (that is, the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the input areas are meshed or belong to a meshed volume, the area(s) can be extruded to a 3D mesh. Note that the *NDIV* argument of the **ESIZE (p. 668)** command should be set before extruding the meshed areas. Alternatively, mesh divisions can be specified directly on the drag line(s) (**LESIZE (p. 1041)**). See the *Modeling and Meshing Guide* for more information.

You can use the **VDRAG (p. 2084)** command to generate 3D interface element meshes for elements INTER194 and INTER195. When generating interface element meshes using **VDRAG (p. 2084)**, you must

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specify the line divisions to generate one interface element directly on the drag line using the **LESIZE (p. 1041)** command. The source area to be extruded becomes the bottom surface of the interface element. Interface elements must be extruded in what will become the element's local x direction, that is, bottom to top.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>Along Lines

---

***VEC**,*Vector*,*Type*,*Method*,*Val1*,*Val2*,*Val3*,*Val4*

Creates a vector.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Vector

Name used to identify the vector. Must be specified.

Type

Vector type:

D --

Double precision real values (default).

Z --

Complex double precision values.

I --

Integer values.

Method

Method used to create the vector:

ALLOC --

Allocate space for a vector (default).

RESIZE --

Resize an existing vector to a new length. Values are kept from the original vector. If the length specified by *Val1* is greater than the original vector length, the additional rows are assigned a value of zero.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

COPY --

Copy an existing vector.

IMPORT --

Import the vector from a file.

LINK --

Link to a column of an existing dense ***DMAT (p. 551)** matrix and use it in subsequent vector calculations. Any changes to the vector are also made to the corresponding matrix column (memory is shared).

Val1*, *Val2*, *Val3*, *Val4*, *Val5

Additional input. The meaning of *Val1* through *Val5* will vary depending on the specified *Method*. See details below.

The following *Valx* field is used with *Method* = ALLOC or *Method* = RESIZE:

Val1

Number of rows in the vector.

The following *Valx* field is used with *Method* = COPY:

Val1

Name of the vector to copy.

Val2

Optional argument to specify either the real or the imaginary part of the values to be copied. This option only applies when copying a complex value vector to a real value vector.

REAL

Copy the real part of the vector to the output vector.

IMAG

Copy the imaginary part of the vector to the output vector.

The following table describes the *Valx* fields used with *Method* = IMPORT.

***Method* = IMPORT**					
Val1	***Val2***	***Val3***	***Val4***	***Val5***	**Description**
FULL	File name [a]	RHS - Load vector GVEC - Constraint equation constant terms	(not used)	(not used)	Import a load vector or nodal mapping vector from an existing FULL file.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method = IMPORT					
Val1	***Val2***	***Val3***	***Val4***	***Val5***	**Description**
		BACK - nodal mapping vector (internal to user) FORWARD - nodal mapping vector (user to internal)			
MODE	File name [a]	Mode number	(not used)	(not used)	Import a mode from an existing MODE file.
RST	File name [a]	Data set number	(not used)	Result Type: NSL - Nodal Solution VSL - Transient velocity solution ASL - Transient acceleration solution RF - Reaction Forces	Import degree of freedom results from an existing RST file.
RFRQ	File name [a]	FRQ - Frequencies of the modal analysis DSP - reduced complex displacement	- Iteration number	(not used)	Import the vector of frequencies of the modal analysis. Import a reduced complex displacement, at a given iteration number.
ANS	File name [a]	Loc - Location in the File	Base - base pointer	(not used)	Import a record in an Ansys File, specifying its location. If Base is not 0, Loc is a relative value from this base pointer.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method = IMPORT					
Val1	***Val2***	***Val3***	***Val4***	***Val5***	**Description**
DMIG	File name [a]	- Separator character (default=blank) - or 'F' means formatted file (see ***DMAT (p. 551)** command notes (p. 556))	if Val3 == 'F', length of the fields.	(not used)	Import a vector (typically a load vector) from an existing Nastran DMIG file.
SUB	File name [a]	RHS - Load vector	(not used)	(not used)	Import a load vector from an existing SUB file.
EMAT	File name [a]	RHS - Load vector	Element number	(not used)	Import an element load vector from an existing EMAT file.
MAT	File name [a]	(not used)	(not used)	(not used)	Restore from a previous ***EXPORT (p. 716)** (*FORMAT* = MAT) command.
APDL	Array parameter name	(not used)	(not used)	(not used)	Import an existing array parameter.
HBMAT	File name [a]	File format: ASCII BINARY	(not used)	(not used)	Import a load vector from an existing Harwell-Boeing format file.

[a] File name is case-sensitive with a 32-character maximum.

The following *Valx* fields are used with *Method* = LINK:

Val1

Name of the ***DMAT (p. 551)** matrix.

Val2

Column number of the matrix to link to.

Notes

Use the ***DMAT (p. 551)** command to create a matrix.

For more information on the BACK and FORWARD nodal mapping vectors, see Degree of Freedom Ordering in the *Ansys Parametric Design Language Guide*.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VEDIT, *Par*

Allows numerical array parameters to be graphically edited.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

Name of the array parameter to be edited.

Notes

Invokes a graphical editing system that displays array parameter values in matrix form, and allows the use of the mouse to edit individual values. The starting array subscripts must be defined, such as ***VEDIT (p. 2089)**,A(4,6,1), to indicate the section of the array to be edited. The array section starts at the specified array element and continues to the maximum extent of the array parameter. Row and column index values may be set or changed in any plane, and those values will be applied to all planes. The menu system must be on (**/MENU (p. 1121)**) when this command is issued. Graphical editing is not available for character array parameters. The ***VEDIT (p. 2089)** command can not be used in a macro or other secondary input file.

This command is not applicable to 4- or 5-D arrays.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Parameters>Define/Edit

VEORIENT, *VNUM*, *Option*, *VALUE1*, *VALUE2*

Specifies brick element orientation for volume mapped (hexahedron) meshing.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VNUM

Number identifying volume for which elements are to be oriented (no default).

Option

Option for defining element orientation:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KP

Orientation is determined by two keypoints on the volume. Input the keypoint numbers (KZ1 and KZ2) in fields *VALUE1* and *VALUE2*, respectively. The element z-axis points from KZ1 toward KZ2. Element x and y directions point away from KZ1 along edges of the volume to make a right-hand triad. (The element x- and y-axes are uniquely determined by this specification.)

LINE

Orientation is determined by one of the lines defining the volume. Input the line number in field *VALUE1*. The element z direction follows the direction of the line. Input a negative value if the desired z direction is opposite to the direction of the specified line. (The element x- and y-axes are uniquely determined by this specification.) (*VALUE2* is not used.)

AREA

Orientation is determined by one of the areas defining the volume. The area represents the desired element top surface. Input the area number as *VALUE1*. The shortest line in the volume connected to the area will be used to specify the element z direction. (If more than one shortest line exists, the lowest numbered of those is used.) Element x and y directions are not uniquely specified by this option. (*VALUE2* is not used.)

THIN

Align the element z normal to the thinnest dimension of the volume. The shortest line in the volume is used to specify the element z direction. (If more than one shortest line exists, the lowest numbered of those is used.) Element x and y directions are not uniquely specified by this option. (*VALUE1* and *VALUE2* are not used.)

DELE

Delete the previously defined volume orientation for the specified volume (*VNUM*). (*VALUE1* and *VALUE2* are not used.)

VALUE1, VALUE2

Parameters required for the element z-axis direction specification. The meaning of *VALUE1* and *VALUE2* will depend on the chosen *Option*. See the description of *Option* above for details.

Command Default

Elements are not oriented in any specific manner.

Notes

Use **VEORIENT (p. 2089)** before the **VMESH (p. 2140)** command to specify the desired orientation of brick elements in a mapped mesh. **VEORIENT (p. 2089)** has no effect on tetrahedron meshes, extruded meshes (**VROTAT (p. 2155)**, **VDRAG (p. 2084)**, **VEXT (p. 2091)**, etc.), or swept meshes (**VSWEEP (p. 2168)**).

Proper brick orientation is essential for certain element types such as SOLID185 Layered Solid, SOLID186 Layered Solid, and SOLSH190. In such cases, use **VEORIENT (p. 2089)** or **EORIENT (p. 646)** to achieve the desired orientation. For other brick element types, you may need to specify element orientation to

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

control orthotropic material property directions without concern for the element connectivity. For those cases, the **ESYS (p. 685)** command is the preferred method of specifying the material property directions.

For *Option* = LINE, AREA, and THIN, the orientation will be internally converted to an equivalent *Option* = KP specification (KP,KZ1,KZ2). Use the **VLIST (p. 2137)** command to view the element orientations (in terms of KZ1 and KZ2) associated with each volume.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Delete Specification

Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z Along Line

Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z by 2 Keypoints

Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z in Thin Direction

Main Menu>Preprocessor>Meshing>Mesh Attributes>Volume Brick Orient>Z Normal to Area

---

VEXT, *NA1*, *NA2*, *NINC*, *DX*, *DY*, *DZ*, *RX*, *RY*, *RZ*

Generates additional volumes by extruding areas.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1*, *NA2*, *NINC

Set of areas (*NA1* to *NA2* in steps of *NINC*) that defines the pattern to be extruded. *NA2* defaults to *NA1*, *NINC* defaults to 1. If *NA1* = ALL, *NA2* and *NINC* are ignored and the pattern is defined by all selected areas. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NA1* (*NA2* and *NINC* are ignored).

DX*, *DY*, *DZ

Increments to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system (*DR*, *D*θ, *DZ* for cylindrical; *DR*, *D*θ, *D*Φ for spherical).

RX*, *RY*, *RZ

Scale factors to be applied to the X, Y, and Z keypoint coordinates in the active coordinate system (*RR*, *R*θ, *RZ* for cylindrical; *RR*, *R*θ, *R*Φ for spherical). Note that the *R*θ and *R*Φ scale factors are interpreted as angular *offsets*. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

Notes

Generates additional volumes (and their corresponding keypoints, lines, and areas) by extruding and scaling a pattern of areas in the active coordinate system.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If element attributes have been associated with the input area via the **AATT (p. 85)** command, the opposite area generated by the **VEXT (p. 2091)** operation will also have those attributes (that is, the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the areas are meshed or belong to meshed volumes, a 3D mesh can be extruded with this command. Note that the *NDIV* argument on the **ESIZE (p. 668)** command should be set before extruding the meshed areas.

Scaling of the input areas, if specified, is performed first, followed by the extrusion.

In a non-Cartesian coordinate system, the **VEXT (p. 2091)** command locates the end face of the volume based on the active coordinate system. However, the extrusion is made along a straight line between the end faces. Note that solid modeling in a toroidal coordinate system is not recommended.

Caution:

Use of the **VEXT (p. 2091)** command can produce unexpected results when operating in a non-Cartesian coordinate system. For a detailed description of the possible problems that may occur, see Solid Modeling in the *Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>By XYZ Offset

*VFACT, *FACTR*, *FACT1*, *FACT2*, *FACT3*

Applies a scale factor to array parameters.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

FACTR

Scale factor applied to results (ParR) parameter. Defaults to 1.0.

FACT1

Scale factor applied to first parameter (Par1 or ParI). Defaults to 1.0.

FACT2

Scale factor applied to second parameter (Par2 or ParJ). Defaults to 1.0.

FACT3

Scale factor applied to third parameter (Par3 or ParK). Defaults to 1.0.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command Default

Use 1.0 for all scale factors.

Notes

Applies a scale factor to parameters used in certain ***VXX*** and ***MXX*** operations. Typical scale factor applications are of the form:

ParR = *FACTR**f(*FACT1**Par1)

or

ParR = *FACTR**((*FACT1**Par1) o (*FACT2**Par2))

The factors are applied to each input parameter value before the operation and to the result value after the operation. The scale factor settings are *reset* to the default (1.0) after each ***VXX*** or ***MXX*** operation. Use ***VSTAT (p. 2167)** to list settings.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

VFCO,*Action*,*ENCL*,*LEVEL*

Controls the use and level of view factor condensation for symmetric radiation.

AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Action to be performed:

DEFINE

Defines the level of view factor condensation (default).

CLEAR

Resets the level of view factor condensation to 0 for all enclosures. All subsequent arguments are ignored.

STATUS

Outputs the *LEVEL* of view factor condensation for each enclosure in the model.

ENCL

Previously defined enclosure number for the view factor adjustment.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LEVEL

Key that controls the level of condensation used in calculating the view factor matrix for models with symmetry. Efficiency gains increase with increasing values of *LEVEL*.

0

View factor condensation is turned off (default). The view factor matrix is calculated for all facets, as described in View Factor Matrix: General in the *Theory Reference*.

1

View factor condensation is turned on. With condensation on, view factors for dependent facets are not calculated, which reduces solution time for models with symmetry. The view factor matrix is calculated only for independent facets as described in View Factor Matrix for a Model with Symmetry in the *Theory Reference*. This option achieves better efficiency than *LEVEL* = 0. Element NMISC data is written to the results file for both independent and dependent facets.***GET (p. 797)**,,RAD,,NETHF, which also uses element fluxes, is based on independent and dependent facets.

2

This option achieves even more efficiency gains than *LEVEL* = 1, but it requires more memory and loses some information. Note that when *LEVEL* = 2 is used, dependent facets are unselected, and no element NMISC data is written to the results file for dependent facets. The same is true for ***GET (p. 797)**,,RAD,,NETHF, which also uses element fluxes and is based on independent facets only.

Notes

If view factor condensation is turned on (**VFCO (p. 2093)**,,*ENCL*,1 or **VFCO (p. 2093)**,,*ENCL*,2):

- The dependent facets do not participate in the solution, and only the independent view factors are calculated as described in View Factor Matrix for a Model with Symmetry in the *Theory Reference*.
- The problem is reduced to solving only for the independent radiosity flux as described in Radiosity Equations Simplified for Models with Symmetry in the *Theory Reference*.
- The **VFSM (p. 2104)** command operates on the condensed view factor matrix.

VFCO (p. 2093) must be issued before the view factors are computed by issuing either **VFOPT (p. 2099)**,NEW or **SOLVE (p. 1822)**.

Example Usage

Example of a 3D Open Enclosure with Symmetry: Radiation Analysis with Condensed View Factor Calculation in the *Thermal Analysis Guide*

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VFDI, *Opt*, *ENCLOSURE*, *DIAGTYPE*, *REALDATA1*, *REALDATA2*

Specifies view factor diagnostic options.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Opt

Diagnostic option:

0

Do not write diagnostic information (default).

1

Write diagnostic information to the database file Jobname.db.

2

Write diagnostic information to a separate view factor diagnosis file Jobname.vfd.

3

Combine options 1 and 2 so that diagnostic information is written in both the Jobname.db and the Jobname.vfd files.

STAT

Print all specfied values for **VFDI (p. 2095)** command arguments (*Opt*, *ENCLOSURE*, *DIAGTYPE*,*REALDATA1*, *REALDATA2*)

ENCLOSURE

Specifies enclosure number to diagnose. (Enclosure number is set using the **SF (p. 1733)**, **SFA (p. 1739)**, **SFE (p. 1761)**, or **SFL (p. 1774)** command with *LAB* = RDSF.)

0

Diagnose all enclosures (default).

>0

Enclosure number to diagnose.

DIAGTYPE

Specifies diagnostic type.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

View factor sum diagnostic. Creates an element component to be written to `Jobname.db` and/or `Jobname.vfd` for rows that have a view factor sum equal to a value that is within the minimum and maximum view factor sum specified by *REALDATA1* and *REALDATA2*, respectively. If the sum is outside of this range, nothing is written. The element component name is "_ENCLOS_*n*", where *n* is the enclosure number.

REALDATA1

For the view factor sum diagnostic (*DIAGTYPE* = 1), this specifies the minimum view factor sum (default = -0.01).

REALDATA2

For the view factor sum diagnostic (*DIAGTYPE* = 1), this specifies the maximum view factor sum (default = 1.1).

Command Default

No diagnostic information is written.

Notes

The **VFDI (p. 2095)** command specifies a view factor diagnostic tool that is valid for radiation analyses using the radiosity solver method. It can be used to provide debugging information on the view factor matrix when you must restart an analysis after an unconverged solution. The command writes information to the `Jobname.db` and/or `Jobname.vfd` files in the working directory that help identify where view factors in a complex model may be causing a solution failure. For more information, see Troubleshooting with View Factor Diagnostics in the *Thermal Analysis Guide*.

Issue **VFDI (p. 2095)** to specify diagnostic information to be written the next time view factors are created and written to the database by a subsequent **VFOPT (p. 2099)**,NEW or **SOLVE (p. 1822)** command.

Example Usage

Using **VFDI** to Write View Factor Diagnostic Information in the *Thermal Analysis Guide*

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VFILL,*ParR*,*Func*,*CON1*,*CON2*,*CON3*,*CON4*,*CON5*,*CON6*,*CON7*,*CON8*,*CON9*,*CON10*

Fills an array parameter.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting numeric array parameter vector. See ***SET (p. 1720)** for name restrictions.

Func

Fill function:

DATA

Assign specified values *CON1*, *CON2*, etc. to successive array elements. Up to 10 assignments may be made at a time. Any *CON* values after a blank *CON* value are ignored.

RAMP

Assign ramp function values: *CON1*+((n-1)**CON2*), where n is the loop number (***VLEN (p. 2136)**). To specify a constant function (no ramp), set *CON2* to zero.

RAND

Assign random number values based on a uniform distribution RAND(*CON1*,*CON2*), where:

CON1 is the lower bound (defaults to 0.0)

CON2 is the upper bound (defaults to 1.0)

GDIS

Assign random sample of Gaussian distributions GDIS(*CON1*,*CON2*) where:

CON1 is the mean (defaults to 0.0)

CON2 is the standard deviation (defaults to 1.0)

TRIA

Assigns random number values based on a triangular distribution TRIA(*CON1*,*CON2*,*CON3*) where:

CON1 is the lower bound (defaults to 0.0)

CON2 is the location of the peak value (*CON1* $\leq$ *CON2* $\leq$*CON3*; *CON2* defaults to 0 if *CON1* $\leq$ 0 $\leq$ *CON3*, *CON1* if 0 $\leq$ *CON1*, or *CON3* if *CON3* $\leq$ 0)

CON3 is the upper bound (defaults to 1.0 + *CON1* if *CON1* $\geq$ 0 or 0.0 if *CON1* $\leq$ 0)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

BETA

Assigns random number values based on a beta distribution BETA(*CON1*,*CON2*,*CON3*,*CON4*) where:

CON1 is the lower bound (defaults to 0.0)

CON2 is the upper bound (defaults to 1.0 + *CON1* if *CON1* $\geq$ 0 or 0.0 if *CON1* $\leq$ 0)

CON3 and *CON4* are the alpha and beta parameters, respectively, of the beta function. Alpha and beta must both be positive; they default to 1.0.

GAMM

Assigns random number values based on a gamma distribution: GAMM(*CON1*,*CON2*,*CON3*) where:

CON1 is the lower bound (defaults to 0.0)

CON2 and *CON3* are the alpha and beta parameters, respectively, of the gamma function. Alpha and beta must both be positive; they default to 1.0.

RIGID

Generates the rigid body modes with respect to the reference point coordinates (*CON1*, *CON2*, *CON3*). The dimensions of the array parameter *ParR* are (dim_1,dim_2) where dim_1 is the maximum node number (including internal nodes but excluding orientation nodes) multiplied by the number of degrees of freedom, and dim_2 is the number of rigid body modes (which corresponds to the number of structural degrees of freedom).

CLUSTER

Generates excitation frequencies with clustering option CLUSTER(*CON1*,*CON2*,*CON3*,*CON4*,*%CON5%*) where:

CON1 is the lower end of the frequency range in Hz (0 < *CON1*)

CON2 is the upper end of the frequency range in Hz (*CON1* < *CON2*)

CON3 is the number of points on each side of the natural frequency (4 $\leq$ *CON3* $\leq$ 20, defaults to 4)

CON4 is the constant damping ratio value or an array parameter (size NFR) specifying the damping ratios (if zero or blank, defaults to constant damping ratio of 0.005)

CON5 is an array parameter (size NFR) specifying the natural frequencies in Hz

The dimension of the resulting array parameter ParR is less than 2+NFR*(2**CON3*+1) where NFR is the number of natural frequencies defined in *CON5*.

CON1*, *CON2*, *CON3*, . . . , *CON10

Constants used with above functions.

Notes

Operates on input data and produces one output array parameter vector according to:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ParR = f(*CON1*, *CON2*, ...)

where the functions (f) are described above. Operations use successive array elements (***VLEN (p. 2136)**, ***VMASK (p. 2139)**) with the default being all successive elements. For example, ***VFILL (p. 2097)**,A,RAMP,1,10 assigns A(1) = 1.0, A(2) = 11.0, A(3) = 21.0, etc. ***VFILL (p. 2097)**,B(5,1),DATA,1.5,3.0 assigns B(5,1) = 1.5 and B(6,1) = 3.0. Absolute values and scale factors may be applied to the result parameter (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). See the ***VOPER (p. 2142)** command for details.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Parameters>Fill

VFOPT,*Opt*,*Filename*,*Ext*,*Dir*,*Filetype*,*Fileformat*,*WRIO*

Specifies options for the view factor file and calculates view factors.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Opt

View factor option:

NEW

Calculate view factors, store them in the database, and write them to a file. This is an action option that is executed immediately when the command is issued. For a coupled-field analysis with **NLGEOM (p. 1231)**,ON, issue either the **UPGEOM (p. 2058)** or **UPCOORD (p. 2056)** command before issuing **VFOPT (p. 2099)**,NEW to calculate view factors based on the updated geometry.

OFF

Do not recalculate or read view factors if they already exist in the database; otherwise calculate them at the next **SOLVE (p. 1822)** command. Remaining arguments are ignored. This option is the default.

READ

If the command is issued in the solution processor (**/SOLU (p. 1821)**), this option reads view factors from the specified binary file when the next **SOLVE (p. 1822)** command is issued. *FileType* must be set to BINA (binary). For subsequent **SOLVE (p. 1822)** commands, the program switches back to the default option (OFF).

If the command is issued in the radiation processor (**/AUX12 (p. 209)**), this option immediately reads view factors from the specified binary file. *FileType* must be set to BINA (binary). The program switches back to the default option (OFF) after reading the view factors.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NONE

Do not write view factors to a file when the next **SOLVE (p. 1822)** command is issued. Remaining arguments are ignored.

Fname

File name for view factor matrix. Default = `Jobname`.

Ext

Filename extension for view factor matrix. Default = `.vf`.

Dir

Directory path for view factor matrix. If you do not specify a directory path, it will default to your working directory.

Filetype

View factor file type:

BINA

Binary (default).

ASCI

ASCII.

Fileformat

Format for the specified *Filetype*:

Binary files (*Filetype* = BINA):

0

No compression. (View factor file size may be very large.)

1

Zeroes are compressed out. (Useful for large models to reduce the view factor file size.)

ASCII files (*Filetype* = ASCI):

0

10F7.4 (low precision, lower accuracy).

1

7F11.8 (high precision, higher accuracy).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WRIO

Write only the view factors of independent facets for symmetric models. This option is valid when view factor condensation is enabled (**VFCO (p. 2093)**,,,1 or **VFCO (p. 2093)**,,,2) and further increases efficiency by reducing read/write time; for more information, see Considerations for Symmetric Models Using View Factor Condensation (p. 2101) below.

ON

Writes only the independent view factors when view factor condensation has been enabled, providing additional efficiency gains. After view factors are computed and written to a file, *WRIO* is automatically reset to OFF.

OFF

Turns off option to write only the independent view factors (default).

Notes

The **VFOPT (p. 2099)** command allows you to deactivate the view factor computation (*Opt* = OFF) if the view factors already exist in the database. The default behavior is OFF upon encountering the second and subsequent **SOLVE (p. 1822)** commands in the solution processor.

The **VFOPT (p. 2099)** command controls view factor calculations and the reading/writing of view factors at the load step level only. It is not supported at the substep level. To update view factors for a coupled-field analysis with **NLGEOM (p. 1231)**,ON at the substep level, you must use the **VFUP (p. 2110)** command.

When *Opt* = READ, only a previously calculated view factor binary file is valid. View factors are read only and are not written after they are read in. Do not issue **VFOPT (p. 2099)**,OFF or **VFOPT (p. 2099)**,NONE until after the next **SOLVE (p. 1822)** command is executed.

If you want to read in view factors after restarting a radiation analysis, issue **VFOPT (p. 2099)**,READ after **ANTYPE (p. 162)**,,REST.

For 3D analyses, two options are available for calculating view factors when running a distributed-memory parallel solution:

- Issue a **SOLVE (p. 1822)** command -- View factors are calculated in parallel mode if no view factors were previously calculated.
- Issue a **VFOPT (p. 2099)**,NEW command -- View factors are calculated in serial mode.

For 2D analyses, view factors are calculated in serial mode.

Considerations for Symmetric Models Using View Factor Condensation

For symmetric models using view factor condensation (**VFCO (p. 2093)**,,,1 or **VFCO (p. 2093)**,,,2), read/write time can be reduced by writing only the independent view factors since they are the only non-zero values in the view factor matrix as described in View Factor Matrix for a Model with Symmetry in the *Theory Reference*.

When view factor condensation is turned off (**VFCO (p. 2093)**,,,0), the full matrix is written to the view factor file when **VFOPT (p. 2099)**,NEW is issued:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\begin{bmatrix} \mathbf{F}_{II} & \mathbf{F}_{ID} \\ \mathbf{F}_{DI} & \mathbf{F}_{DD} \end{bmatrix}$ (full matrix) where the view factor matrix is decomposed, and the subscripts, I and D, denote independent and dependent. (See View Factor Matrix: General in the *Theory Reference* for details.)

When view factor condensation is turned on (**VFCO (p. 2093)**,,,1 or **VFCO (p. 2093)**,,,2), use *WRIO* to control what is written to the view factor file:

- When **VFOPT (p. 2099)**,NEW,,,,,OFF is issued, the lumped matrix and zeros are written:
  $\begin{bmatrix} \mathbf{F}^L & 0 \\ 0 & 0 \end{bmatrix}$ (lumped matrix and zeros) (For the definition of $\mathbf{F}^L$, see Equation 6.21 in the *Theory Reference*.)
- When **VFOPT (p. 2099)**,NEW,,,,,ON is issued, only the lumped matrix is written:
  $\mathbf{F}^L$ (lumped matrix only) .

Example Usage

2D Radiation Analysis Using the Radiosity Method with Decimation and Symmetry

3D Open Enclosure with Symmetry: Radiation Analysis with Condensed View Factor Calculation

Menu Paths

Main Menu>Preprocessor>Radiation Opts>View Factor

Main Menu>Radiation Opt>Radiosity Meth>Compute

Main Menu>Radiation Opt>Radiosity Meth>View Factor

Main Menu>Solution>Radiation Opts>View Factor

VFQUERY,*SRCELEM*,*TARELEM*,*--*,*WrOpt*

Queries and prints element Hemicube view factors and average view factor.

AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SRCELEM

Elements representing the source radiating surfaces used to query the view factor at the target element(s). If *SRCELEM* = P, graphical picking is enabled (valid only in the GUI). If *SRCELEM* = ALL, all selected elements will have their view factors queried. A component name may also be substituted for *SRCELEM*. Selected elements must be flagged for surface to surface radiation in order to query view factors (**SF (p. 1733)**, **SFA (p. 1739)**, or **SFE (p. 1761)** with Lab = RDSF). The view factors must have been previously computed.

TARELEM

Element for view factor query. If *TARELEM* = P, graphical picking is enabled (valid only in the GUI). If *TARELEM* = ALL, all selected elements will have their view factors queried. A component name may also be substituted for *TARELEM*. Selected elements must be flagged for surface to surface

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

radiation in order to query view factors (**SF (p. 1733)**, **SFA (p. 1739)**, or **SFE (p. 1761)** with Lab = RDSF). The view factors must have been previously computed.

--

Unused field.

WrOpt

Option to write view factors to the output file (`Jobname.out`).

WON

Write view factors to the output file (default).

WOFF

Do not write view factors to the output file. This is useful for reducing the size of the output file.

Notes

This is an action command that:

- Queries the view factors for the interactions between each selected source element with each selected target element and computes the average view factor.
- Writes the view factors for the interactions between each source element with each target element to the output file if *WrOpt* = WON.

To retrieve the calculated average view factor, issue ***GET (p. 797)**,*Par*,RAD,,VFAVG.

If you are only interested in the average view factor, set *WrOpt* = WOFF, and issue ***GET (p. 797)** to reduce the size of the output file and save time.

When resuming a database, issue the command **VFOPT (p. 2099)**,READ before issuing the **VFQUERY (p. 2102)** command.

Menu Paths

Main Menu>Radiation Opt>Radiosity Meth>Query

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VFSM,*Action*,*ENCL*,*OPT*,*MAXITER*,*CONV*

Adjusts view factor matrix to satisfy reciprocity and/or row sum properties (view factor smoothing).

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Action to be performed:

DEFINE

Define a view factor summation (default)

CLEAR

Resets the scaling method to 0 for all enclosures. All subsequent arguments are ignored.

STATUS

Outputs the *OPT* value for each enclosure in the model.

ENCL

Previously defined enclosure number for the view factor adjustment.

OPT

Option key:

0

The view factor matrix values are not adjusted (default).

1

The view factor matrix values are adjusted so that the row sum equals 1.0.

2

The view factor matrix values are adjusted so that the row sum equals 1.0 and the reciprocity relationship is satisfied.

3

The view factor matrix values are adjusted so that the reciprocity relationship is satisfied.

4

The view factor matrix values are adjusted so that the original row sum is maintained and the reciprocity relationship is satisfied.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MAXITER

Maximum number of iterations to achieve convergence. Valid only when *OPT* = 2 or 4. Default is 100.

CONV

Convergence value for row sum. Iterations will continue (up to *MAXITER*) until the maximum residual over all the rows is less than this value. Valid only when *OPT* = 2 or 4. Default is 1E-3.

Command Default

Although the command is not required for solution, appropriate view factor smoothing settings on **VFSM (p. 2104)** are recommended to ensure a good energy balance. If the command is not issued, no view factor adjustments are made, and view factors might not satisfy the row sum and reciprocity relationships.

Notes

To have a good energy balance, it is important to satisfy both the row sum and reciprocity relationships. For more information, see View Factors in the Mechanical APDL Theory Reference.

OPT = 1 and 2 are suitable for perfect enclosures. *OPT* = 1 is less expensive than *OPT* = 2 because no iterations are involved. However, with *OPT* = 1, the reciprocity relationship is not satisfied.

OPT = 3 and 4 are suitable for leaky enclosures. *OPT* = 3 is less expensive than *OPT* = 4 because no iterations are involved. However, with *OPT* = 3, the original row sum is not maintained.

The **VFSM (p. 2104)** command must be used before **VFOPT (p. 2099)** is issued, or Solve is initiated.

Note:

While the primary purpose of the **VFSM (p. 2104)** command is to adjust the viewfactor matrix to satisfy reciprocity and rowsum properties, a side effect of this command is that the model could flip from being an imperfect to a perfect enclosure and the space node ignored if the rowsum becomes 1.0. The program's check for an imperfect enclosure is not based geometry, but rather on the value of the rowsum of all rows of the enclosure view factor matrix. A rowsum close to 1.0 is deemed a perfect enclosure; otherwise, it is an imperfect enclosure, which requires you to define a spacenode. It is important to be aware that the **VFSM (p. 2104)** command can affect the view factor rowsum and potentially also whether the enclosure is treated as an imperfect or perfect enclosure.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VFUN,*ParR*,*Func*,*Par1*,*CON1*,*CON2*,*CON3*

Performs a function on a single array parameter.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting numeric array parameter vector. See ***SET (p. 1720)** for name restrictions.

Func

Function to be performed:

ACOS --

Arccosine: ACOS(*Par1*).

ASIN --

Arcsine: ASIN(*Par1*).

ASORT --

Par1 is sorted in ascending order. ***VCOL (p. 2077)**, ***VMASK (p. 2139)**, ***VCUM (p. 2080)**, and ***VLEN (p. 2136)**,,NINC do not apply. ***VLEN (p. 2136)**,NROW does apply.

ATAN --

Arctangent: ATAN(*Par1*).

COMP --

Compress: Selectively compresses data set. "True" (***VMASK (p. 2139)**) values of *Par1* (or row positions to be considered according to the *NINC* value on the ***VLEN (p. 2136)** command) are written in compressed form to *ParR*, starting at the specified position.

COPY --

Copy: *Par1* copied to *ParR*.

COS --

Cosine: COS(*Par1*).

COSH --

Hyperbolic cosine: COSH(*Par1*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DIRCOS --

Direction cosines of the principal stresses (*n*X9). *Par1* contains the *n*X6 component stresses for the *n* locations of the calculations.

DSORT --

Par1 is sorted in descending order. ***VCOL (p. 2077)**, ***VMASK (p. 2139)**, ***VCUM (p. 2080)**, and ***VLEN (p. 2136)**,,NINC do not apply. ***VLEN (p. 2136)**,NROW does apply.

EULER --

Euler angles of the principal stresses (*n*X3). *Par1* contains the *n*X6 component stresses for the *n* locations of the calculations.

EXP --

Exponential: EXP(*Par1*).

EXPA --

Expand: Reverse of the COMP function. All elements of *Par1* (starting at the position specified) are written in expanded form to corresponding "true" (***VMASK (p. 2139)**) positions (or row positions to be considered according to the *NINC* value on the ***VLEN (p. 2136)** command) of *ParR*.

LOG --

Natural logarithm: LOG(*Par1*).

LOG10 --

Common logarithm: LOG10(*Par1*).

NINT --

Nearest integer: 2.783 becomes 3.0, -1.75 becomes -2.0.

NOT --

Logical complement: values ≤ 0.0 (false) become 1.0 (true). Values > 0.0 (true) become 0.0 (false).

PRIN --

Principal stresses (*n*X5). *Par1* contains the *n*X6 component stresses for the *n* locations of the calculations.

PWR --

Power function: *Par1****CON1*. Exponentiation of any negative number in the vector *Par1* to a non-integer power is performed by exponentiating the positive number and prepending the minus sign. For example, -4**2.3 is -(4**2.3).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SIN --

Sine: SIN(*Par1*).

SINH --

Hyperbolic sine: SINH(*Par1*).

SQRT --

Square root: SQRT(*Par1*).

TAN --

Tangent: TAN(*Par1*).

TANH --

Hyperbolic tangent: TANH(*Par1*).

TANG --

Tangent to a path at a point: the slope at a point is determined by linear interpolation half way between the previous and next points. Points are assumed to be in the global Cartesian coordinate system. Path points are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the x, y, and z coordinate locations, respectively, of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3). The tangent result, *ParR*, must also have 3 consecutive columns of data and will contain the tangent direction vector (normalized to 1.0); such as 1,0,0 for an x-direction vector.

NORM --

Normal to a path and an input vector at a point: determined from the cross-product of the calculated tangent vector (see TANG) and the input direction vector (with the i, j, and k components input as *CON1*, *CON2*, and *CON3*). Points are assumed to be in the global Cartesian coordinate system. Path points are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the x, y, and z coordinate locations, respectively, of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3). The normal result, *ParR*, must also have 3 consecutive columns of data and will contain the normal direction vector (normalized to 1.0); such as 1,0,0 for an x-direction vector.

LOCAL --

Transforms global Cartesian coordinates of a point to the coordinates of a specified system: points to be transformed are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the x, y, and z global Cartesian coordinate locations, respectively, of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3). Results are transformed to coordinate system *CON1* (which may be any valid coordinate system number,

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

such as 1,2,11,12, etc.). The transformed result, *ParR*, must also have 3 consecutive columns of data and will contain the corresponding transformed coordinate locations.

GLOBAL --

Transforms specified coordinates of a point to global Cartesian coordinates: points to be transformed are specified in array *Par1* (having 3 consecutive columns of data, with the columns containing the local coordinate locations (x, y, z or r, θ, z or etc.) of the points). Only the starting row index and the column index for the x coordinates are specified, such as A(1,1). The y and z coordinates (or θ and z, or etc.) of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3). Local coordinate locations are assumed to be in coordinate system *CON1* (which may be any valid coordinate system number, such as 1,2,11,12, etc.). The transformed result, *ParR*, must also have 3 consecutive columns of data, with the columns containing the global Cartesian x, y, and z coordinate locations, respectively.

Par1

Array parameter vector in the operation.

CON1*, *CON2*, *CON3

Constants (used only with the PWR, NORM, LOCAL, and GLOBAL functions).

Notes

Operates on one input array parameter vector and produces one output array parameter vector according to:

ParR = f(*Par1*)

where the functions (f) are described below. Functions are based on the standard FORTRAN definitions where possible. Out-of-range function results (or results with exponents whose magnitudes are approximately greater than 32 or less than -32) produce a zero value. Input and output for angular functions may be radians (default) or degrees (***AFUN (p. 83)**). *ParR* may be the same as *Par1*. Starting array element numbers must be defined for each array parameter vector if it does not start at the first location. For example, ***VFUN (p. 2106)**,A,SQRT,B(5) takes the square root of the fifth element of B and stores the result in the first element of A. Operations continue on successive array elements (***VLEN (p. 2136)**, ***VMASK (p. 2139)**) with the default being all successive elements. Absolute values and scale factors may be applied to all parameters (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). Skipping array elements via ***VMASK (p. 2139)** or ***VLEN (p. 2136)** for the TANG and NORM functions skips only the writing of the results (skipped array element data are used in all calculations). See the ***VOPER (p. 2142)** command for details.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Vector Functions

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VFUP,*Action*,*Opt*,*OPT2*,*OPT3*

Controls view factor updating at the substep level for a coupled-field radiation analysis that includes large-deflection effects.

SOLUTION (p. 38): Radiosity (p. 50)
AUX12 (p. 62): Radiosity Solver (p. 62)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Action to be performed:

DEFINE

Define view factor updating parameters (default).

STAT

Print all specified values for **VFUP (p. 2110)** command arguments.

DEFA

Specifies default values for all arguments, equivalent to issuing **VFUP (p. 2110)**,DEFINE,OFF,1,0.

Opt

Option to enable view factor updates at the substep level or specify their frequency.

ON

Enable view factor updates at the substep *OPT2* and pass *OPT3* frequencies.

OFF

Do not update view factors at the substep level (default). *OPT2* and *OPT3* are ignored.

FRQU

Key to specify a desired value for the substep frequency (*OPT2*) and the pass frequency (*OPT3*). For guidelines on setting these values and example command usage, see Controlling the frequency of view factor updates (p. 2111).

OPT2

Valid only if *Opt* = FRQU. Substep frequency for view factor updates. This value must be a positive integer (defaults to 1). If *OPT2* = 3, view factors are updated every third substep.

OPT3

Valid only if *Opt* = FRQU. Valid values are -1, 0, or a positive integer (defaults to 0).

If *OPT3* = 0, view factors are not updated in the radiation multipass loop.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If *OPT3* is a positive integer, it specifies the pass frequency for view factor updates in the multipass loop.

If *OPT3* = -1, view factors are updated only in the first pass of the multipass loop in the substeps that view factors are being updated (specified by *OPT2*).

Command Default

Although the command is not required for solution, if you do not issue **VFUP (p. 2110)**, view factors are not updated at the substep level. Consequently, results of thermal radiation modeling may be inaccurate because they do not account for changes in the view factors of radiation surfaces as they deform or move.

Notes

Deformation and/or motion of radiating surfaces during an anlysis can cause inaccuracies in radiation calculations if the view factors are not updated to account for these changes. Issue **VFUP (p. 2110)**,DEFINE,ON to enable view factor updates at the substep level and improve simulation accuracy for an analysis that includes all of the following conditions:

- Thermal radiation modeling using the radiosity solver method.
- Coupled-field elements (PLANE222, PLANE223, SOLID225, SOLID226, or SOLID227) having at least structural and thermal degrees of freedom (DOFs).
- Large-deflection effects (**NLGEOM (p. 1231)**,ON).

When view factor updating is activated (*Action* = ON), radiation facet areas are recalculated along with the view factors at the specified substep (*OPT2*) and multipass (*OPT3*) frequencies.

Updates occur only if all of these conditions are met:

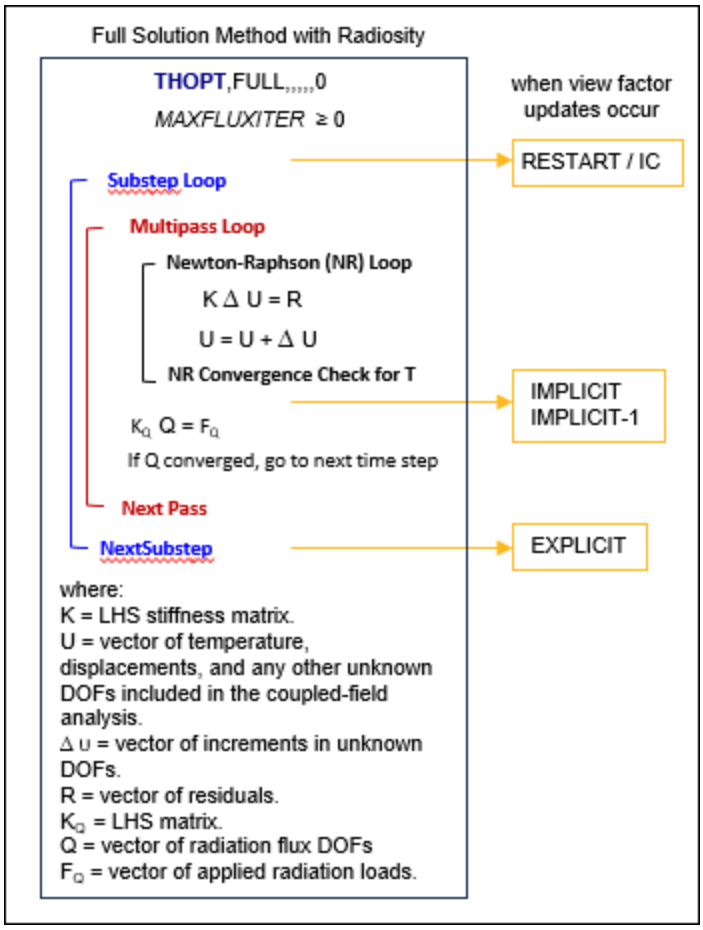
- **VFUP (p. 2110)**,DEFINE,ON has been issued.
- **NLGEOM (p. 1231)**,ON has been issued.
- You have specified KEYOPT(1) = 1 for any surface elements (SURF251 or SURF252) that have been generated via the **RSURF (p. 1631)** command.

Controlling the frequency of view factor updates

The full solution method (**THOPT (p. 1997)**,FULL) with radiosity has three nested loops, which occur for each load step: the substep (or time step) loop, the multipass loop, and the Newton-Raphson (NR) loop. Displacements and temperatures, in addition to any other DOFs that may be included in your coupled-field analysis, are computed in the NR loop. The view factors / radiation areas can be recalculated at different points during the solution, as depicted in Figure 28: View Factor Update Timing (p. 2112) and described in the following discussion.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 28: View Factor Update Timing



As seen in the figure, implicit updates occur within the multipass loop, and updated view factors are used in the current substep. Instead, explicit updates occur at the end of the time step, and updated view factors are used in the next time step. An additional update occurs before entering the substep loop for a restart (**ANTYPE (p. 162)**,, RESTART) or if you have imposed initial conditions on the displacements (**IC (p. 921)**) as indicated.

There is a trade-off between accuracy and computational cost as updating the view factors more often improves accuracy but also increases the solution time. You can control the timing of view factor updates by specifying the substep (*OPT2*) and pass (*OPT3*) frequency, following one of these strategies:

- **Explicit** - View factors are updated at the end of a substep after a converged coupled-field solution is achieved, and the updated view factors are used in the next time step. The value specified for the substep frequency, *OPT2*, determines which substeps include view factor updating. By default, *OPT2* = 1, and view factors are updated at the end of every substep.

For example, if you issue:	View factors are updated
VFUP (p. 2110),DEFINE,FRQU,3,0	at the end of substeps 3,6,9,...

  This is the most stable strategy. If you have convergence difficulties, explicit updates are recommended. For an accurate solution, the substep size must be small. Explicit updates may lead to an energy imbalance in the radiosity solution since the view factor facet areas were calculated

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the preceding time step. For further details, see the discussion related to Figure 30: View Factor Updates and Reciprocity (p. 2115).

- **Implicit** - View factors are updated within the radiation multipass loop, after the Newton-Raphson (RP) loop has produced a converged coupled-field solution and are used in the current pass and substep. The value of the pass frequency, *OPT3*, determines which passes will update the view factors. To have implicit view factor updating, you must set *OPT3* to something other than its default value of zero.

For example, if you issue:	View factors are updated
VFUP (p. 2110),DEFINE,FRQU,5,3	in passes 3,6,9,... for substeps 5,10,15...

  Implicit updates are useful when you have strong nonlinearities, strong structural-thermal coupling, or rapid changes in surface geometry. However, implicit updates can be computationally expensive as view factors are recalculated multiple times within a single substep. Also, Implicit updates of view factors can lead to instabilities and convergence difficulties.

- **Implicit-1** - View factors are updated in the substeps prescribed by the value of the substep frequency, *OPT2*, but they are updated only in the first pass of the radiation multipass loop. To have implicit-1 view factor updating, specify -1 for *OPT3*.

For example, if you issue:	View factors are updated
VFUP (p. 2110), DEFINE, FRQU, 5, -1	only in pass 1 for substeps 5,10,15...

  Use this strategy when the motion of bodies in your model is completely specified, as in rigid body motion (translation and rotation). It is most efficient and accurate to update the view factors in the first pass only when the NR loop achieves a converged structural solution in the first pass and the structural solution does not change for subsequent passes.

Restrictions/Limitations

Enclosure status change and penetration must be avoided

Simulations using the **VFUP (p. 2110)** to update view factors at the substep level are invalid if an enclosure status changes or the surface of a radiating body penetrates the surface of another radiating body. Figure 29: Examples of Radiation Problems that Undergo an Enclosure Status Change (p. 2114) illustrates some examples of enclosure status changes that must be avoided. For example, if two radiating bodies in close proximity penetrate each other due to thermal expansion during the analysis, their view factors become zero, and results using the **VFUP (p. 2110)** command to update view factors are unpredictable and inaccurate. It is better to model bodies within close proximity using thermal contact radiation (contact and target elements: CONTA172, CONTA174, CONTA175, CONTA178,TARGE169, and TARGE170).

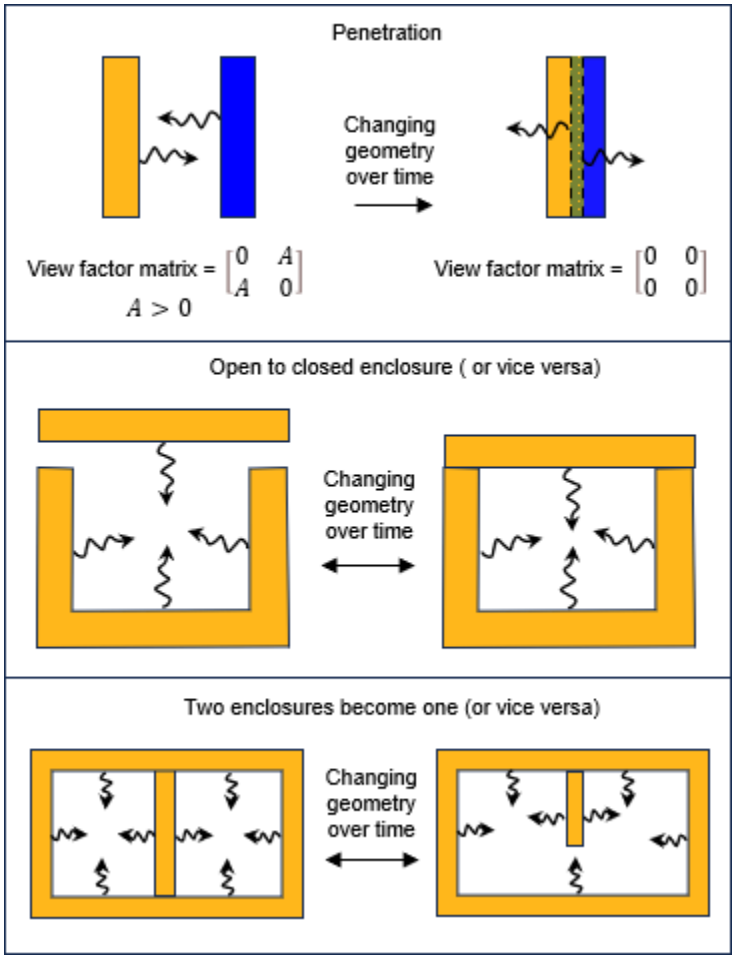
You must also be careful to avoid an enclosure status change, for example if movement or deformation in the analysis causes an open enclosure to become closed or vice versa. If an enclosure is undergoing a status change in your analysis, split the load step into multiple load steps such that:

- There is no status change in the individual load steps.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **VFSM (p. 2104)** settings are appropriate for the enclosure status in individual load steps.

Figure 29: Examples of Radiation Problems that Undergo an Enclosure Status Change

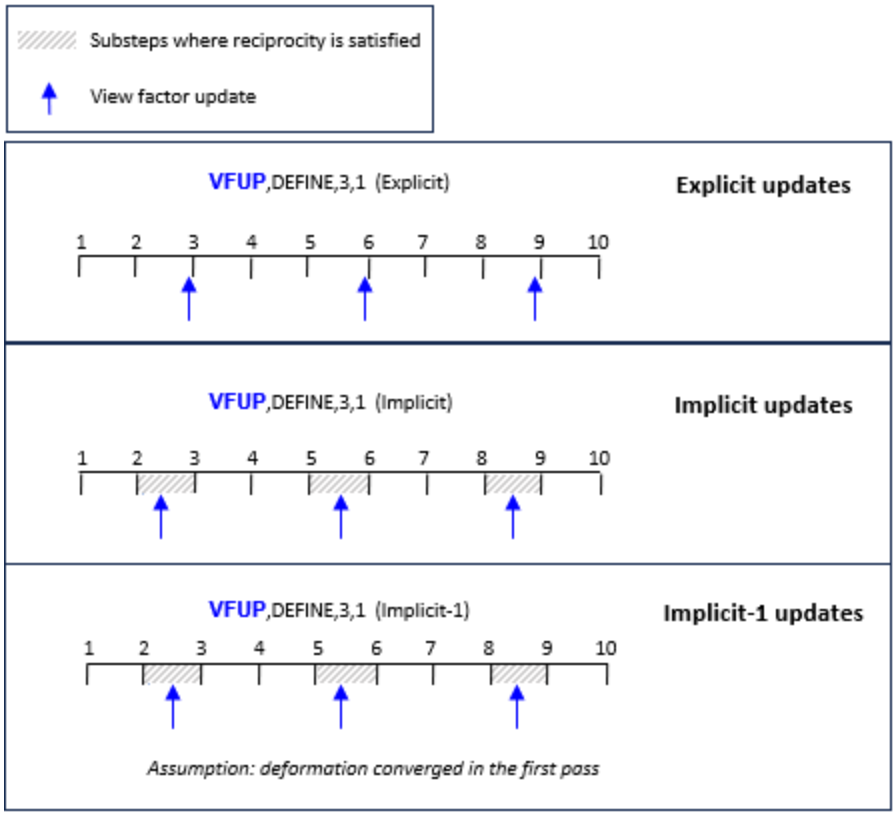


View factor Updates and Reciprocity

For explicit updates, view factors are recalculated at the end of the substep and applied in the next substep. Consequently, they violate the reciprocity rule in the next substep as the radiating areas change since the view factors were calculated from the areas of the previous substep. This may lead to energy imbalance issues. The same problem exists if you specify a substep (*OPT2*) or pass (*OPT3*) frequency other than one. Figure 30: View Factor Updates and Reciprocity (p. 2115) illustrates this issue for different updating strategies. Reciprocity / row sum properties are only satisfied in substeps where view factors are updated in every pass (*OPT3* = 1) provided you have used the **VFSM (p. 2104)** command appropriately (see View Factors in the *Theory Reference*).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30: View Factor Updates and Reciprocity



Supported/restricted commands at the substep level with VFUP (p. 2110),DEFINE,ON

The following commands are not supported at the substep level:

- **VFOPT (p. 2099)**
- **THOPT (p. 1997)**,QUASI
- **VFDI (p. 2095)**

The following commands may be used to query and print view factors or average view factors at the substep level:

- View factor queries (**VFQUERY (p. 2102)**) are based on the last update.
- Any ***GET (p. 797)** command with *Entity* = RAD is based on the last view factor update. To retrieve the latest average view factor, issue ***GET (p. 797)**,*Par*,RAD,0,VFAVG.
- To print view factors to the output file at the substep level, issue **MPRINT (p. 1178)** before **SOLVE (p. 1822)**.

The following radiation commands are valid at the substep level, and may be used with view factor updating enabled (**VFUP (p. 2110)**,DEFINE,ON):

- **HEMIOPT (p. 904)**, **RADOPT (p. 1567)**, **V2DOPT (p. 2070)**, **SPCTEMP (p. 1826)**, **SPCNOD (p. 1825)**, and **VFCO (p. 2093)**

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Settings on the **VFSM (p. 2104)** command to adjust view factors (view factor smoothing) for different enclosures are valid for all substeps. However, if there is an enclosure status change during a load step, **VFSM (p. 2104)** settings may become inappropriate and results inaccurate. See earlier discussion on inaccurate results if motion or deformation causes an enclosure status change (Figure 29: Examples of Radiation Problems that Undergo an Enclosure Status Change (p. 2114)). To model a situation where deformation changes the enclosure status, you must solve the problem in multiple load steps and change **VFSM (p. 2104)** settings in each load step to accurately model the state of the enclosure.

Example Usage

Example of a Structural-Thermal Analysis with Substep-Level View Factor Updating in the *Thermal Analysis Guide*

VM321 in the *Ansys Mechanical APDL Verification Manual*

Menu Paths

This command cannot be accessed from a menu.

VGEN, *ITIME*, *NV1*, *NV2*, *NINC*, *DX*, *DY*, *DZ*, *KINC*, *NOELEM*, *IMOVE*

Generates additional volumes from a pattern of volumes.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

ITIME

Do this generation operation a total of *ITIME*s, incrementing all keypoints in the given pattern automatically (or by *KINC*) each time after the first. *ITIME* must be > 1 for generation to occur.

NV1*, *NV2*, *NINC

Generate volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes (**VSEL (p. 2163)**). If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

DX*, *DY*, *DZ

Keypoint location increments in the active coordinate system (--, Dθ, DZ for cylindrical, --, Dθ, -- for spherical).

KINC

Keypoint increment between generated sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NOELEM

Specifies if elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original volumes, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether to redefine the existing volumes:

0

Generate additional volumes as requested with the *ITIME* argument.

1

Move original volumes to new position retaining the same keypoint line, and area numbers (*ITIME*, *KINC*, and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates additional volumes (and their corresponding keypoints, lines, areas and mesh) from a given volume pattern. The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings of the pointers. End slopes of the generated lines remain the same (in the active coordinate system) as those of the given pattern. For example, radial slopes remain radial, etc. Generations which produce volumes of a size or shape different from the pattern (that is, radial generations in cylindrical systems, radial and phi generations in spherical systems, and theta generations in elliptical systems) are not allowed. Note that solid modeling in a toroidal coordinate system is not recommended. Volume, area, and line numbers are automatically assigned (beginning with the lowest available values (**NUMSTR (p. 1301)**)).

Menu Paths

Main Menu>Preprocessor>Modeling>Copy>Volumes

Main Menu>Preprocessor>Modeling>Move / Modify>Volumes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VGET,*ParR*,*Entity*,*ENTNUM*,*Item1*,*IT1NUM*,*Item2*,*IT2NUM*,*KLOOP*

Retrieves values and stores them into an array parameter.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting vector array parameter. See ***SET (p. 1720)** for name restrictions. The program creates the array parameter if it does not exist.

Entity

Entity keyword. Valid keywords are NODE, ELEM, KP, LINE, AREA, VOLU, etc. as shown for *Entity* = in the tables below.

ENTNUM

The number of the entity (as shown for *ENTNUM* = in the tables below).

Item1

The name of a particular item for the given entity. Valid items are as shown in the *Item1* columns of the tables below.

IT1NUM

The number (or label) for the specified *Item1* (if any). Valid *IT1NUM* values are as shown in the *IT1NUM* columns of the tables below. Some *Item1* labels do not require an *IT1NUM* value.

Item2,IT2NUM

A second set of item labels and numbers to further qualify the item for which data is to be retrieved. Most items do not require this level of information.

KLOOP

Field to be looped on:

0 or 2 --

Loop on the *ENTNUM* field (default).

3 --

Loop on the *Item1* field.

4 --

Loop on the *IT1NUM* field. Successive items are as shown with *IT1NUM*.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 --

Loop on the *Item2* field.

6 --

Loop on the *IT2NUM* field. Successive items are as shown with *IT2NUM*.

Notes

Retrieves values for specified items and stores the values in an output vector of a user-named array parameter according to:

ParR = f(*Entity*, *ENTNUM*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*)

where (f) is the ***GET (p. 797)** function; *Entity*, *Item1*, and *Item2* are keywords; and *ENTNUM*, *IT1NUM*, and *IT2NUM* are numbers or labels corresponding to the keywords. Looping continues over successive entity numbers (*ENTNUM*) for the *KLOOP* default. For example, ***VGET (p. 2118)**,A(1),ELEM,5,CENT,X returns the centroid x-location of element 5 and stores the result in the first location of A. Retrieving continues with element 6, 7, 8, etc., regardless of whether the element exists or is selected, until successive array locations are filled. Use ***VLEN (p. 2136)** or ***VMASK (p. 2139)** to skip locations. Absolute values and scale factors may be applied to the result parameter (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). See the ***VOPER (p. 2142)** command for general details. Results can be put back into an analysis by writing a file of the desired input commands with the ***VWRITE (p. 2174)** command. See also the ***VPUT (p. 2149)** command.

Both ***GET (p. 797)** and ***VGET (p. 2118)** retrieve information from the active data stored in memory. The database is often the source, and sometimes the information is retrieved from common memory blocks that Mechanical APDL uses to manipulate information. Although POST1 and POST26 operations use a `*.rst` file, GET data is accessed from the database or from the common blocks. Get operations do not access the `*.rst` file directly.

The ***VGET (p. 2118)** command retrieves both the unprocessed real and the imaginary parts (original and duplicate sector nodes and elements) of a cyclic symmetry solution.

Each of the sections for accessing *VGET parameters are shown in the following order:

- *VGET PREP7 Items (p. 2119)
- *VGET POST1 Items (p. 2123)

This command is valid in any processor.

*VGET PREP7 Items

- Table 233: *VGET PREP7 Items, Entity = NODE (p. 2120)
- Table 234: *VGET PREP7 Items, Entity = ELEM (p. 2120)
- Table 235: *VGET PREP7 Items, Entity = KP (p. 2121)
- Table 236: *VGET PREP7 Items, Entity = LINE (p. 2121)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Table 237: *VGET PREP7 Items, Entity = AREA (p. 2121)
- Table 238: *VGET PREP7 Items, Entity = VOLU (p. 2122)
- Table 239: *VGET PREP7 Items, Entity = CDSY (p. 2122)
- Table 240: *VGET PREP7 Items, Entity = RCON (p. 2123)
- Table 241: *VGET PREP7 Items, Entity = TLAB (p. 2123)

Table 233: *VGET PREP7 Items, Entity = NODE

***Entity* = NODE, *ENTNUM* = *n* (node number)**		
VGET (p. 2118)*, *ParR*, NODE, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system
ANG	XY, YZ, ZX	THXY, THYZ, THZX rotation angle
NSEL		Select status of node *n*: -1 = unselected, 0 = undefined, 1 = selected
NLIST		Returns the list of selected nodes (*ENTNUM* is ignored)

Table 234: *VGET PREP7 Items, Entity = ELEM

***Entity* = ELEM, *ENTNUM* = *n* (element number)**		
VGET (p. 2118)*, *ParR*, ELEM, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
NODE	1, 2, ... 20	Node number at position 1, 2, ... 20 of element *n*
CENT	X, Y, Z	Centroid X, Y, or Z location (based on shape function) in the active coordinate system
ADJ	1, 2, ... 6	Number of elements adjacent to face 1, 2, ... 6
ATTR	*name*	Number assigned to specified attribute; *name* = MAT, TYPE, REAL, ESYS, ENAM, or SECN
GEOM		Characteristic element geometry. Length of line element (straight line between ends), area of area element, or volume of volume element. Issuing ***VGET (p. 2118)** for an element returns a signed value. To always get a positive value, issue ***VABS (p. 2072)**,1 prior to issuing the ***VGET (p. 2118)** command.
ESEL		Select status of element *n*: -1 = unselected, 0 = undefined, 1 = selected
SHPAR	*Test*	Element shape test result for selected element *n*, where *Test* = ASPE (aspect ratio), JACR (Jacobian ratio), MAXA (maximum corner angle), PARA (deviation from parallelism of opposite edges), or WARP (warping factor)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = ELEM, *ENTNUM* = *n* (element number)		
VGET (p. 2118), *ParR*, ELEM, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
ELIST		Returns the list of selected elements (*ENTNUM* is ignored)

Table 235: *VGET PREP7 Items, Entity = KP

Entity = KP, *ENTNUM* = *n* (keypoint number)		
VGET (p. 2118), *ParR*, KP, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
LOC	X, Y, Z	X, Y, or Z location in the active coordinate system
ATTR	*name*	Number assigned to specified attribute; *name* = MAT, TYPE, REAL, ESYS, NODE, or ELEM
DIV		Divisions (element size setting) from **KESIZE (p. 978)**
KSEL		Select status of keypoint *n*: -1 = unselected, 0 = undefined, 1 = selected
KLIST		Returns the list of selected keypoints (*ENTNUM* is ignored)

Table 236: *VGET PREP7 Items, Entity = LINE

Entity = LINE, *ENTNUM* = *n* (line number)		
VGET (p. 2118), *ParR*, LINE, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
KP	1, 2	Keypoint number at position 1 or 2
ATTR	*name*	Number assigned to specified attribute; *name* = MAT, TYPE, REAL, ESYS, NNOD, NELM, or NDIV (NNOD = number of nodes, NELM = number of elements, NDIV = number of divisions)
LENG		Length
LSEL		Select status of line *n*: -1 = unselected, 0 = undefined, 1 = selected
LLIST		Returns the list of selected lines (*ENTNUM* is ignored)

Table 237: *VGET PREP7 Items, Entity = AREA

Entity = AREA, *ENTNUM* = *n* (area number)				
VGET (p. 2118), *ParR*, AREA, *n*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*, *KLOOP				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
LOOP	1,2, ...	LINE	1, 2, ... *p*	IT1NUM is the loop number, and must be input if LINE is to be retrieved. IT2NUM is line number at position 1, 2, ... *p*
ATTR	*name*			Number assigned to specified attribute; *name* = MAT, TYPE, REAL, ESYS, SECN, NNOD, or NELM (NNOD = number of nodes, NELM = number of elements)
AREA	-			Area (after last **ASUM (p. 203)**)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = AREA, *ENTNUM* = *n* (area number)				
VGET (p. 2118), *ParR*, AREA, *n*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*, *KLOOP				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
ASEL	-			Select status of area *n*: -1 = unselected, 0 = undefined, 1 = selected
ALIST	-			Returns the list of selected areas (*ENTNUM* is ignored)

Table 238: *VGET PREP7 Items, Entity = VOLU

Entity = VOLU, *ENTNUM* = *n* (volume number)				
VGET (p. 2118), *ParR*, VOLU, *n*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*, *KLOOP				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
SHELL	1, 2, ...	AREA	1, 2, ... *p*	IT1NUM is the shell number, and must be input if AREA is to be retrieved. IT2NUM is area number at position 1, 2 ... *p*
ATTR	*name*			Number assigned to specified attribute; *name* = MAT, TYPE, REAL, ESYS, NNOD, or NELM (NNOD = number of nodes, NELM = number of elements)
VOLU	-			Volume (after last **VSUM (p. 2167)**)
VSEL	-			Select status of volume *n*: -1 = unselected, 0 = undefined, 1 = selected
VLIST	-			Returns the list of selected volumes (*ENTNUM* is ignored)

Table 239: *VGET PREP7 Items, Entity = CDSY

Entity = CDSY, *ENTNUM* = *n* (coordinate system number)		
VGET (p. 2118), *ParR*, CDSY, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
LOC	X, Y, Z	X, Y, or Z origin location (global Cartesian coordinate)
ANG	XY, YZ, ZX	THXY, THYZ, or THZX rotation angle (°) relative to the global Cartesian coordinate system

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = CDSY, *ENTNUM* = *n* (coordinate system number)		
VGET (p. 2118), *ParR*, CDSY, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
ATTR	*name*	Number assigned to specified attribute; *name* = KCS, KTHET, KPHI, PAR1, or PAR2. If the coordinate system is undefined, KCS returns as -1.0

Table 240: *VGET PREP7 Items, Entity = RCON

Entity = RCON, *ENTNUM* = *n* (real constant set number)		
VGET (p. 2118), *ParR*, RCON, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
CONST	1, 2, ... *m*	Real constant value for constant 1, 2, ... *m*

Table 241: *VGET PREP7 Items, Entity = TLAB

Entity = *TLAB*, *ENTNUM* = *n* (*TLAB* is the *Lab* data table label on the TB (p. 1899) command. *n* is the material number.)				
VGET (p. 2118), *ParR*, *TLAB*, *n*, *Item1*, *IT1NUM*, , , *KLOOP				
Item1	**IT1NUM**	**Item2**	**IT2NUM**	**Description**
TEMP	*val*	CONST	*num*	*value* is the temperature value at which to retrieve table data. *num* is the constant number whose value is to be retrieved. For constants input as X, Y points, the constant numbers are consecutive with the X constants being the odd numbers, beginning with one.

*VGET POST1 Items

- Table 242: *VGET POST1 Items, Entity = CYCCALC (p. 2124)
- Table 243: *VGET POST1 Items, Entity = ELEM (p. 2125)
- Table 244: *VGET POST1 Items, Entity = MEMBER (p. 2125)
- Table 245: *VGET POST1 Items, Entity = NODE, Element Nodal Results (p. 2125)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Table 246: *VGET POST1 Items, Entity = NODE, Nodal Degree of Freedom Results (p. 2129)

Note:

Vector items are in the active results coordinate system unless otherwise specified.

Table 242: *VGET POST1 Items, Entity = CYCCALC

***Entity* = CYCCALC, *ENTNUM* = *n* (CYCSPEC (p. 475) specification number)**				
VGET (p. 2118), *ParR*, CYCCALC, *n*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*, *KLOOP				
Item1	**IT1NUM**	**Item2**	**IT2NUm**	**Description**
FREQ	*Frequency point*	SECTOR	*sector*	**CYCSPEC (p. 475)** results at the IT1NUM frequency or sector (depending on *KLOOP*)
		SECMAX		**CYCSPEC (p. 475)** maximum results
The frequency point refers to the harmonic solution data set number (*NSET* on the **SET (p. 1724)** command). For *KLOOP* = 4 or SECMAX, returns the requested sector results for all frequencies and *IT1NUM* is ignored. For *KLOOP* = 6, returns the requested frequency results for all sectors and *IT2NUM* is ignored.				

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = CYCCALC, *ENTNUM* = *n* (CYCSPEC (p. 475) specification number)				
*VGET (p. 2118), *ParR*, CYCCALC, *n*, *Item1*, *IT1NUM*, *Item2*, *IT2NUM*, *KLOOP*				
Item1	**IT1NUM**	**Item2**	**IT2NUm**	**Description**
All other *KLOOP* options are invalid.				

Table 243: *VGET POST1 Items, Entity = ELEM

Entity = ELEM, *ENTNUM* = *n* (element number)		
*VGET (p. 2118), *ParR*, ELEM, *n*, *Item1*, *IT1NUM*, , , *KLOOP*		
Item1	**IT1NUM**	**Description**
ETAB	*Label*	Any user-defined element table label (see **ETABLE (p. 687)** command)

Table 244: *VGET POST1 Items, Entity = MEMBER

Entity = MEMBER, *ENTNUM* = *N* (GroupID)		
*VGET (p. 2118),*Par*,MEMBER,*N*, *Item1*, *IT1NUM*, , , *KLOOP*		
Item1	**IT1NUM**	**Description**
TEMP	MIN, MAX	Minimum or maximum temperature of members (individual reinforcings) with GroupID = *N* in the selected set of reinforcing elements

Table 245: *VGET POST1 Items, Entity = NODE, Element Nodal Results

Entity = NODE, *ENTNUM* = *n* (node number)			
*VGET (p. 2118), *ParR*, NODE, *n*, *Item1*, *IT1NUM*, , , *KLOOP*			
Item1	**IT1NUM**	**Description**	**Item2**
S	X, Y, Z, XY, YZ, XZ	Component stress	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal stress	
	INT, EQV	Stress intensity or equivalent stress	
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR)	
	1, 2, 3	Principal total strain	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= NODE,** ***ENTNUM*** **=** ***n*** **(node number)**			
VGET (p. 2118),** ***ParR**, NODE,** ***n*****,** ***Item1*****,** ***IT1NUM*****, , ,** ***KLOOP***			
Item1	**IT1NUM**	**Description**	**Item2**
	INT, EQV	Total strain intensity or total equivalent strain	
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal elastic strain	
	INT, EQV	Elastic strain intensity or elastic equivalent strain	
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal plastic strain	
	INT, EQV	Plastic strain intensity or plastic equivalent strain	
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results.
	1, 2, 3	Principal creep strain	
	INT, EQV	Creep strain intensity or creep equivalent strain	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = NODE, *ENTNUM* = *n* (node number)			
***VGET (p. 2118)**, *ParR*, NODE, *n*, *Item1*, *IT1NUM*, , , *KLOOP*			
Item1	**IT1NUM**	**Description**	**Item2**
			ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
	1, 2, 3	Principal thermal strain	
	INT, EQV	Thermal strain intensity or thermal equivalent strain	
EPSW		Swelling strain	*Item2* controls whether nodal-averaged results are used. Valid labels are: AUTO - Use nodal-averaged results, if available. Otherwise use element-based results. ESOL- Use element-based results only. NAR - Use nodal-averaged results only.
EPDI	X, Y, Z, XY, YZ, XZ	Component diffusion strain	
	1, 2, 3	Principal diffusion strain	
	INT, EQV	Diffusion strain intensity or diffusion equivalent strain	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = NODE, *ENTNUM* = *n* (node number)			
VGET (p. 2118), *ParR*, NODE, *n*, *Item1*, *IT1NUM*, , , *KLOOP			
Item1	**IT1NUM**	**Description**	**Item2**
NL	SEPL	Equivalent stress (from stress-strain curve)	
	SRAT	Stress state ratio	
	HPRES	Hydrostatic pressure	
	EPEQ	Accumulated equivalent plastic strain	
	CREQ	Accumulated equivalent creep strain.	
	PSV	Plastic state variable	
	PLWK	Plastic work/volume	
HS	X, Y, Z	Component magnetic field intensity from current sources (in the global Cartesian coordinate system)	
BFE	TEMP	Body temperatures (calculated from applied temperatures) as used in solution	
TG	X, Y, Z, SUM	Component thermal gradient and sum	
TF	X, Y, Z, SUM	Component thermal flux and sum	
PG	X, Y, Z, SUM	Component pressure gradient and sum	
EF	X, Y, Z, SUM	Component electric field and sum	
D	X, Y, Z, SUM	Component electric flux density and sum	
H	X, Y, Z, SUM	Component magnetic field intensity and sum	
B	X, Y, Z, SUM	Component magnetic flux density and sum	
FMAG	X, Y, Z, SUM	Component electromagnetic force and sum	
CONT	STAT	Contact status	
	PENE	Contact penetration	
	PRES	Contact pressure	
	SFRIC	Contact friction stress	
	STOT	Contact total stress (pressure plus friction)	
	SLIDE	Contact sliding distance	
	GAP	Contact gap distance	
	FLUX	Total heat flux at contact surface	

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity = NODE, *ENTNUM* = *n* (node number)			
VGET (p. 2118), *ParR*, NODE, *n*, *Item1*, *IT1NUM*, , , *KLOOP			
Item1	**IT1NUM**	**Description**	**Item2**
	CNOS	Total number of contact status changes during substep	
	FPRS	Actual applied fluid penetration pressure	
Element nodal results are the average nodal value of the selected elements			

Table 246: *VGET POST1 Items, Entity = NODE, Nodal Degree of Freedom Results

Entity = NODE, *ENTNUM* = *n* (node number)		
VGET (p. 2118), *ParR*, NODE, *n*, *Item1*, *IT1NUM*, , , *KLOOP		
Item1	**IT1NUM**	**Description**
U	X, Y, Z	X, Y, or Z structural displacement
ROT	X, Y, Z	X, Y, or Z structural rotation
TEMP		Temperature. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use TBOT, TE2, TE3, ..., TTOP instead of TEMP. Alternative get functions: TEMP(N), TBOT(N), TE2(N), etc.
PRES		Pressure
VOLT		Electric potential
MAG		Magnetic scalar potential
V	X, Y, Z	X, Y, or Z fluid velocity; X, Y, or Z nodal velocity in a transient structural analysis (analysis with **ANTYPE (p. 162)**,TRANS)
A	X, Y, Z	X, Y, or Z magnetic vector potential; X, Y or Z nodal acceleration in a transient structural analysis (analysis with **ANTYPE (p. 162)**,TRANS)
CURR		Current
EMF		Electromotive force drop

Menu Paths

Utility Menu>Parameters>Get Array Data

VGET,*Par*,*IR*,*TSTRT*,*KCPLX*

Moves a variable into an array parameter vector.

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Par

Array parameter vector in the operation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

IR

Reference number of the variable (1 to NV (**NUMVAR (p. 1302)**)).

TSTRT

Time (or frequency) corresponding to start of *IR* data. If between values, the nearer value is used.

KCPLX

Complex number key:

0

Use the real part of the *IR* data.

1

Use the imaginary part of the *IR* data.

Notes

Moves a variable into an array parameter vector. The starting array element number must be defined. For example, **VGET (p. 2129)**,A(1),2 moves variable 2 (starting at time 0.0) to array parameter A. Looping continues from array element A(1) with the index number incremented by one until the variable is filled. The number of loops may be controlled with the ***VLEN (p. 2136)** (p. 2136) command (except that loop skipping (*NINC*) is not allowed). For multi-dimensioned array parameters, only the first (row) subscript is incremented.

Menu Paths

Main Menu>TimeHist Postpro>Table Operations>Variable to Par

VGLUE, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Generates new volumes by "gluing" volumes.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1, NV2, NV3, . . . , NV9

Numbers of the volumes to be glued. If *NV1* = ALL, all selected volumes will be glued (*NV2* to *NV9* will be ignored). If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Use of the **VGLUE (p. 2130)** command generates new volumes by "gluing" input volumes. The glue operation redefines the input volumes so that they share areas along their common boundaries. The new volumes encompass the same geometry as the original volumes. This operation is only valid if the inter-

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sections of the input volumes are areas along the boundaries of those volumes. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

The **VGLUE (p. 2130)** command results in the merging of areas, lines, and keypoints at the common volume boundaries. The areas, lines, and keypoints of the lower numbered volume will be kept. This means one must be aware of volume numbering when multiple **VGLUE (p. 2130)** commands are applied to avoid any "ungluing" of geometry.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Glue>Volumes

---

/VIEW, *WN*, *XV*, *YV*, *ZV*

Defines the viewing direction for the display.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

XV, YV, ZV

The object is viewed along the line from point *XV*,*YV*,*ZV* (in the global coordinate system) to the global coordinate system origin. For section displays, the cutting plane is assumed to be perpendicular to this line. If *XV* = WP, modify view to be normal to the currently defined working plane. Defaults to (0,0,1).

Command Default

0,0,1 view. The default reference orientation is X-axis horizontal to the right, Y-axis vertical upward, and Z-axis out from the screen (normal). See **/VUP (p. 2173)** command to change reference orientation.

Notes

The view line is always normal to the screen. The view is selected by defining a point (in the global Cartesian coordinate system) representing a point along the viewing line. This point, and the global Cartesian coordinate system origin, define the line along which the object is viewed while looking toward the origin. Any point along the view line may be used, that is, (1,1,1) and (2,2,2) give the same view. The display orientation may be changed as desired (**/ANGLE (p. 140)**). The display coordinate system type may be changed (from Cartesian to cylindrical, spherical, toroidal, etc.) with the **DSYS (p. 582)** command.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>Path Operations>Define Path>On Working Plane

Main Menu>Preprocessor>Path Operations>Define Path>On Working Plane

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Utility Menu>PlotCtrls>View Settings>Viewing Direction

VIMP, *VOL*, *CHGBND*, *IMPLEVEL*

Improves the quality of the tetrahedral elements in the selected volume(s).

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VOL

Number of the volume containing the tetrahedral elements to be improved. If *VOL* = ALL (default), improve the tetrahedral elements in all selected volumes. If *VOL* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *VOL*.

CHGBND

Specifies whether to allow boundary modification. Boundary modification includes such things as changes in the connectivity of the element faces on the boundary and the addition of boundary nodes. (Also see "Notes" (p. 2133) below for important usage information for *CHGBND*.)

0

Do not allow boundary modification.

1

Allow boundary modification (default).

IMPLEVEL

Identifies the level of improvement to be performed on the elements. (Improvement occurs primarily through the use of face swapping and node smoothing techniques.)

0

Perform the least amount of swapping/smoothing.

1

Perform an intermediate amount of swapping/smoothing.

2

Perform the greatest amount of swapping/smoothing.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3

Perform the greatest amount of swapping/smoothing, plus additional improvement techniques (default).

Notes

VIMP (p. 2132) is useful for further improving a volume mesh created in Mechanical APDL (**VMESH (p. 2140)**), especially quadratic tetrahedral element meshes.

The command enables you to improve a given tetrahedral mesh by reducing the number of poorly-shaped tetrahedral elements (in particular, the number of sliver tetrahedral elements)--as well as the overall number of elements--in the mesh. It also improves the overall quality of the mesh.

Regardless of the value of the *CHGBND* argument, boundary mid-nodes can be moved.

When loads or constraints have been placed on boundary nodes or mid-nodes, and boundary mid-nodes are later moved, the program issues a warning message indicating that it will not update the loads or constraints.

Even when *CHGBND* = 1, no boundary modification is performed on areas and lines that are not modifiable. For example, areas that are adjacent to other volumes or that contain shell elements, or lines that are not incident on modifiable areas, contain beam elements, or have line divisions specified for them (**LESIZE (p. 1041)**).

Menu Paths

Main Menu>Preprocessor>Meshing>Modify Mesh>Improve Tets>Volumes

---

VINP, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Finds the pairwise intersection of volumes.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1, NV2, NV3, . . . , NV9

Numbers of volumes to be intersected pairwise. If *NV1* = ALL, *NV2* to *NV9* are ignored and the pairwise intersection of all selected volumes is found. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Finds the pairwise intersection of volumes. The pairwise intersection is defined as all regions shared by any two or more volumes listed on this command. New volumes will be generated where the original volumes intersect pairwise. If the regions of pairwise intersection are only areas, new areas will be generated. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Pairwise>Volumes

---

VINV, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Finds the intersection of volumes.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1, NV2, NV3, . . . , NV9

Numbers of volumes to be intersected. If *NV1* = ALL, *NV2* to *NV9* are ignored, and the intersection of all selected volumes is found. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Finds the common (not pairwise) intersection of volumes. The common intersection is defined as the regions shared (in common) by **all** volumes listed on this command. New volumes will be generated where the original volumes intersect. If the regions of intersection are only areas, new areas will be generated instead. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Intersect>Common>Volumes

---

*VITRP, *ParR*, *ParT*, *ParI*, *ParJ*, *ParK*

Forms an array parameter by interpolation of a table.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting array parameter. See ***SET (p. 1720)** for name restrictions.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ParT

The name of the TABLE array parameter. The parameter must exist as a dimensioned array of type TABLE (***DIM (p. 530)**).

ParI

Array parameter vector of I (row) index values for interpolation in *ParT*.

ParJ

Array parameter vector of J (column) index values for interpolation in *ParT* (which must be at least 2D).

ParK

Array parameter vector of K (depth) index values for interpolation in *ParT* (which must be 3D).

Notes

Forms an array parameter (of type ARRAY) by interpolating values of an array parameter (of type TABLE) at specified table index locations according to:

ParR = f(*ParT*, *Parl*, *ParJ*, *ParK*)

where *ParT* is the type TABLE array parameter, and *ParI*, *ParJ*, *ParK* are the type ARRAY array parameter vectors of index values for interpolation in *ParT*. See the ***DIM (p. 530)** command for TABLE and ARRAY declaration types. Linear interpolation is used. The starting array element number for the TABLE array (*ParT*) is not used (but a value must be input). Starting array element numbers must be defined for each array parameter vector if it does not start at the first location. For example, ***VITRP (p. 2134)**,R(5),TAB(1,1),X(2),Y(4) uses the second element of X and the fourth element of Y as index values (row and column) for a 2D interpolation in TAB and stores the result in the fifth element of R. Operations continue on successive array elements (***VLEN (p. 2136)**, ***VMASK (p. 2139)**) with the default being all successive elements. Absolute values and scale factors may be applied to the result parameter (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). See the ***VOPER (p. 2142)** command for details.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Vector Interpolate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***VLEN**, *NROW*, *NINC*

Specifies the number of rows to be used in array parameter operations.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

NROW

Number of rows to be used with the ***V*XX** or ***M*XX** operations. Defaults to the number of rows needed to fill the result array.

NINC

Perform the operation on every *NINC* row (defaults to 1).

Command Default

Fill all locations of the result array from the specified starting location.

Notes

Specifies the number of rows to be used in array parameter operations. The size of the submatrix used is determined from the upper left starting array element (defined on the operation command) to the lower right array element (defined by the number of rows on this command and the number of columns on the ***VCOL (p. 2077)** command). *NINC* allows skipping row operations for some operation commands. Skipped rows are included in the row count. The starting row number must be defined on the operation command for each parameter read and for the result written.

The default *NROW* is calculated from the maximum number of rows of the result array (the ***DIM (p. 530)** row dimension) minus the starting location + 1. For example, ***DIM (p. 530)**,R,,10 and a starting location of R(7) gives a default of 4 loops (filling R(7), R(8), R(9), and R(10)). Repeat operations automatically terminate at the last row of the result array. Existing values in the rows and columns of the results matrix remain unchanged where not overwritten by the requested input or operation values.

The stride (*NINC*) allows operations to be performed at regular intervals. It has no effect on the total number of row operations. Skipped operations retain the previous result. For example, ***DIM (p. 530)**,R,,6, with a starting location of R(1), *NROW* = 10, and *NINC* = 2 calculates values for locations R(1), R(3), and R(5) and retains values for locations R(2), R(4), and R(6). A more general skip control may be done by masking (***VMASK (p. 2139)**). The row control settings are *reset* to the defaults after each ***V*XX** or ***M*XX** operation. Use ***VSTAT (p. 2167)** to list settings.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VLIST, *NV1*, *NV2*, *NINC*

Lists the defined volumes.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1*, *NV2*, *NINC

List volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* are ignored and all selected volumes (**VSEL (p. 2163)**) are listed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

An attribute (TYPE, MAT, REAL, or ESYS) listed as a zero is unassigned; one listed as a positive value indicates that the attribute was assigned with the **VATT (p. 2075)** command (and will not be reset to zero if the mesh is cleared); one listed as a negative value indicates that the attribute was assigned using the attribute pointer [TYPE, MAT, REAL, or ESYS] that was active during meshing (and will be reset to zero if the mesh is cleared). A "-1" in the "nodes" column indicates that the volume has been meshed but there are no interior nodes. The volume size is listed only if a **VSUM (p. 2167)** command has been performed on the volume. Volume orientation attributes (KZ1 and KZ2) are listed only if a **VEORIENT (p. 2089)** command was previously used to define an orientation for the volume.

This command is valid in any processor.

Menu Paths

Utility Menu>List>Volumes

VLSCALE, *NV1*, *NV2*, *NINC*, *RX*, *RY*, *RZ*, *KINC*, *NOELEM*, *IMOVE*

Generates a scaled set of volumes from a pattern of volumes.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1*, *NV2*, *NINC

Set of volumes (*NV1* to *NV2* in steps of *NINC*) that defines the pattern to be scaled. *NV2* defaults to *NV1*, *NINC* defaults to 1. If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is defined by all selected volumes. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

RX*, *RY*, *RZ

Scale factors to be applied to the X, Y, and Z keypoint coordinates in active coordinate system (*RR*, *R*θ, *RZ* for cylindrical; *RR*, *R*θ, *R*Φ for spherical). Note that the *R*θ and *R*Φ scale factors are interpreted

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as angular *offsets*. For example, if CSYS = 1, *RX*, *RY*, *RZ* input of (1.5,10,3) would scale the specified keypoints 1.5 times in the radial and 3 times in the Z direction, while adding an offset of 10 degrees to the keypoints. Zero, blank, or negative scale factor values are assumed to be 1.0. Zero or blank angular offsets have no effect.

KINC

Increment to be applied to keypoint numbers for generated set. If zero, the lowest available keypoint numbers will be assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Nodes and elements associated with the original volumes will be generated (scaled) if they exist.

1

Nodes and elements will *not* be generated.

IMOVE

Specifies whether volumes will be moved or newly defined:

0

Additional volumes will be generated.

1

Original volumes will be *moved* to new position (*KINC* and *NOELEM* are ignored). Use only if the old volumes are no longer needed at their original positions. Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a scaled set of volumes (and their corresponding keypoints, lines, areas, and mesh) from a pattern of volumes. The MAT, TYPE, REAL, and ESYS attributes are based on the volumes in the pattern and not the current settings. Scaling is done in the active coordinate system. Volumes in the pattern could have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Scale>Volumes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***VMASK**, *Par*

Specifies an array parameter as a masking vector.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par

Name of the mask parameter. The starting subscript must also be specified.

Command Default

No mask parameter specified (use true for all operations).

Notes

Specifies the name of the parameter whose values are to be checked for each resulting row operation. The mask vector usually contains only 0 (for false) and 1 (for true) values. For each row operation the corresponding mask vector value is checked. A true value allows the operation to be done. A false value skips the operation (and retains the previous results). A mask vector can be created from direct input, such as M(1) = 1,0,0,1,1,0,1; or from the DATA function of the ***VFILL (p. 2097)** command. The NOT function of the ***VFUN (p. 2106)** command can be used to reverse the logical sense of the mask vector. The logical compare operations (LT, LE, EQ, NE, GE, and GT) of the ***VOPER (p. 2142)** command also produce a mask vector by operating on two other vectors. Any numeric vector can be used as a mask vector since the actual interpretation assumes values less than 0.0 are 0.0 (false) and values greater than 0.0 are 1.0 (true). If the mask vector is not specified (or has fewer values than the result vector), true (1.0) values are assumed for the unspecified values. Another skip control may be input with *NINC* on the ***VLEN (p. 2136)** command. If both are present, operations occur only when both are true. The mask setting is *reset* to the default (no mask) after each ***V*XX** or ***M*XX** operation. Use ***VSTAT (p. 2167)** to list settings.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Operation Settings

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VMESH,*NV1*,*NV2*,*NINC*

Generates nodes and volume elements within volumes.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1*, *NV2*, *NINC

Mesh volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and all selected volumes (**VSEL (p. 2163)**) are meshed. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

Notes

Missing nodes required for the generated elements are created and assigned the lowest available numbers (**NUMSTR (p. 1301)**). During a batch run and if elements already exist, a mesh abort will write an alternative database file (`File.dbe`) for possible recovery.

Tetrahedral mesh expansion (**MOPT (p. 1151)**,TETEXPND,*Value*) is supported for both the **VMESH (p. 2140)** and **FVMESH (p. 784)** commands.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Free

Main Menu>Preprocessor>Meshing>Mesh>Volumes>Mapped>4 to 6 sided

VOFFST,*NAREA*,*DIST*,*KINC*

Generates a volume, offset from a given area.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NAREA

Area from which generated volume is to be offset. If *NAREA* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

DIST

Distance normal to given area at which keypoints for generated volume are to be located. Positive normal is determined from the right-hand rule keypoint order.

KINC

Increment to be applied to the keypoint numbers between sets. If zero, keypoint numbers will be automatically assigned beginning with the lowest available value (**NUMSTR (p. 1301)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates a volume (and its corresponding keypoints, lines, and areas) by offsetting from an area. The direction of the offset varies with the given area normal. End slopes of the generated lines remain the same as those of the given pattern.

If element attributes have been associated with the input area via the **AATT (p. 85)** command, the opposite area generated by the **VOFFST (p. 2140)** operation will also have those attributes (that is, the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the areas are meshed or belong to meshed volumes, a 3D mesh can be extruded with this command. Note that the *NDIV* argument on the **ESIZE (p. 668)** command should be set before extruding the meshed areas.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>Along Normal

---

VOLUMES

Specifies "Volumes" as the subsequent status topic.

PREP7 (p. 22): Status (p. 35)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This is a status (**STAT (p. 1861)**) topic command. Status topic commands are generated by the GUI and appear in the log file (`Jobname.log`) if status is requested for some items by choosing **Utility Menu> List> Status**. This command will be immediately followed by a **STAT (p. 1861)** command, which will report the status for the specified topic.

If entered directly into the program, the **STAT (p. 1861)** command should immediately follow this command.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VOPER,*ParR*,*Par1*,*Oper*,*Par2*,*CON1*,*CON2*

Operates on two array parameters.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting array parameter vector. See ***SET (p. 1720)** for name restrictions.

Par1

First array parameter vector in the operation. May also be a scalar parameter or a literal constant.

Oper

Operations:

ADD--

Addition: *Par1*+*Par2*.

SUB --

Subtraction: *Par1*-*Par2*.

MULT --

Multiplication: *Par1***Par2*.

DIV --

Division: *Par1*/*Par2* (a divide by zero results in a value of zero).

MIN --

Minimum: minimum of *Par1* and *Par2*.

MAX --

Maximum: maximum of *Par1* and *Par2*.

LT --

Less than comparison: *Par1*<*Par2* gives 1.0 if true, 0.0 if false.

LE --

Less than or equal comparison: *Par1* ≤ *Par2* gives 1.0 if true, 0.0 if false.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EQ --

Equal comparison: *Par1* = *Par2* gives 1.0 if true, 0.0 if false.

NE --

Not equal comparison: *Par1* ≠ *Par2* gives 1.0 if true, 0.0 if false.

GE --

Greater than or equal comparison: *Par1* ≥ Par2 gives 1.0 if true, 0.0 if false.

GT --

Greater than comparison: *Par1*>*Par2* gives 1.0 if true, 0.0 if false.

DER1 --

First derivative: d(*Par1*)/d(*Par2*). The derivative at a point is determined over points half way between the previous and next points (by linear interpolation). *Par1* must be a function (a unique *Par1* value for each *Par2* value) and *Par2* must be in ascending order.

DER2 --

Second derivative: d^2(*Par1*)/d(*Par2*)^2. See also DER1.

INT1 --

Single integral: $\int$ *Par1* d(*Par2*), where *CON1* is the integration constant. The integral at a point is determined by using the single integration procedure described in the *Mechanical APDL Theory Reference*.

INT2 --

Double integral: $\int\int$ *Par1* d(*Par2*), where *CON1* is the integration constant of the first integral and *CON2* is the integration constant of the second integral. If *Par1* contains acceleration data, *CON1* is the initial velocity and *CON2* is the initial displacement. See also INT1.

DOT --

Dot product: *Par1* . *Par2*. *Par1* and *Par2* must each have three consecutive columns of data, with the columns containing the i, j, and k vector components, respectively. Only the starting row index and the column index for the i components are specified for *Par1* and *Par2*, such as A(1,1). The j and k components of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3).

CROSS --

Cross product: *Par1* x *Par2*. *Par1*, *Par2*, and *ParR* must each have 3 components, respectively. Only the starting row index and the column index for the i components are specified for *Par1*, *Par2*, and *ParR*, such as A(1,1). The j and k components of the vector are assumed to begin in the corresponding next columns, such as A(1,2) and A(1,3).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GATH --

Gather: For a vector of position numbers, *Par2*, copy the value of *Par1* at each position number to *ParR*. Example: for *Par1* = 10,20,30,40 and *Par2* = 2,4,1; *ParR* = 20,40,10.

SCAT --

Scatter: Opposite of GATH operation. For a vector of position numbers, *Par2*, copy the value of *Par1* to that position number in *ParR*. Example: for *Par1* = 10,20,30,40,50 and *Par2* = 2,1,0,5,3; *ParR* = 20,10,50,0,40.

ATN2 --

Arctangent: arctangent of *Par1*/*Par2* with the sign of each component considered.

LOCAL --

Transform the data in *Par1* from the global Cartesian coordinate system to the local coordinate system given in *CON1*. *Par1* must be an *N x 3* (that is, vector) or an *N x 6* (that is, stress or strain tensor) array. If the local coordinate system is a cylindrical, spherical, or toroidal system, then you must provide the global Cartesian coordinates in *Par2* as an *N x 3* array. Set *CON2* = 1 if the data is strain data.

GLOBAL --

Transform the data in *Par1* from the local coordinate system given in *CON1* to the global Cartesian coordinate system. *Par1* must be an *N x 3* (that is, vector) or an *N x 6* (that is, stress or strain tensor) array. If the local coordinate system is a cylindrical, spherical, or toroidal system, then you must provide the global Cartesian coordinates in *Par2* as an *N x 3* array. Set *CON2* = 1 if the data is strain data.

Par2

Second array parameter vector in the operation. May also be a scalar parameter or a literal constant.

CON1

First constant (used only with the INT1 and INT2 operations).

CON2

Second constant (used only with the INT2 operation).

Notes

Operates on two input array parameter vectors and produces one output array parameter vector according to:

ParR = *Par1* o *Par2*

where the operations (o) are described below. *ParR* can be the same as *Par1* or *Par2*.

Absolute values and scale factors can be applied to all parameters (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results can be cumulative (***VCUM (p. 2080)**).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Starting array element numbers must be defined for each array parameter vector if it does not start at the first location, such as ***VOPER (p. 2142)**,A,B(5),ADD,C(3) which adds the third element of C to the fifth element of B and stores the result in the first element of A.

Operations continue on successive array elements (***VLEN (p. 2136)**, ***VMASK (p. 2139)**) with the default being all successive elements.

Skipping array elements via ***VMASK (p. 2139)** or ***VLEN (p. 2136)** for the DER_ and INT_ functions skips only the writing of the results (skipped array element data are used in all calculations).

Parameter functions and operations are available to operate on a scalar parameter or a single element of an array parameter, such as SQRT(B) or SQRT(A(4)). (See ***SET (p. 1720)** for more information.)

Operations on a sequence of array elements can be performed by repeating the desired function or operation in a do-loop (***DO (p. 567)**). The vector operations within Mechanical APDLm (***V*XX** commands) are internally programmed do-loops that conveniently perform the indicated operation over a sequence of array elements. If the array is multidimensional, only the first subscript is incremented in the do-loop; that is, the operation repeats in column vector fashion down the array. For example, for A(1,5), A(2,5), A(3,5), etc. The starting location of the row index must be defined for each parameter read and for the result written.

The default number of loops is from the starting result location to the last result location and can be altered via ***VLEN (p. 2136)**.

A logical mask vector can be defined to control at which locations the operations are to be skipped (***VMASK (p. 2139)**). The default is to skip no locations.

Repeat operations automatically terminate at the last array element of the result array column if the number of loops is undefined or if it exceeds the last result array element.

Zeroes are used in operations for values read beyond the last array element of an input array column. Existing values in the rows and columns of the results matrix remain unchanged where not changed by the requested operation values.

The result array column may be the same as the input array column, as results in progress are stored in a temporary array until being moved to the results array at the end of the operation. Results may be overwritten or accumulated with the existing results (***VCUM (p. 2080)**). The default is to overwrite results.

The absolute value can be used for each parameter read or written (***VABS (p. 2072)**). A scale factor (defaulting to 1.0) is also applied to each parameter read and written (***VFACT (p. 2092)**).

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Vector Operations

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VOVLAP, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Overlaps volumes.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1, NV2, NV3, . . . , NV9

Numbers of volumes to be operated on. If *NV1* = ALL, *NV2* to *NV9* are ignored and all selected volumes are used. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Overlaps volumes. Generates new volumes which encompass the geometry of all the input volumes. The new volumes are defined by the regions of intersection of the input volumes, and by the complementary (non-intersecting) regions. See the *Modeling and Meshing Guide* for an illustration. This operation is only valid when the region of intersection is a volume. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Overlap>Volumes

***VPLOT**, *ParX*, *ParY*, *Y2*, *Y3*, *Y4*, *Y5*, *Y6*, *Y7*, *Y8*

Graphs columns (vectors) of array parameters.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParX

Name of the array parameter whose column vector values will be the abscissa of the graph. If blank, row subscript numbers are used instead. *ParX* is not sorted by the program.

ParY

Name of the array parameter whose column vector values will be graphed against the *ParX* values.

Y2, Y3, Y4, . . . , Y8

Additional column subscript of the *ParY* array parameter whose values are to be graphed against the *ParX* values.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

The column to be graphed and the starting row for each array parameter must be specified as subscripts. Additional columns of the *ParY* array parameter may be graphed by specifying column numbers for *Y2*, *Y3*, ...,*Y8*. For example, ***VPLOT (p. 2146)**,TIME (4,6), DISP (8,1),2,3 specifies that the 1st, 2nd, and 3rd columns of array parameter DISP (all starting at row 8) are to be graphed against the 6th column of array parameter TIME (starting at row 4). The columns are graphed from the starting row to their maximum extent. See the ***VLEN (p. 2136)** and ***VMASK (p. 2139)** commands to limit or skip data to be graphed. The array parameters specified on the ***VPLOT (p. 2146)** command must be of the same type (type ARRAY or TABLE; (***DIM (p. 530)**). Arrays of type TABLE are graphed as continuous curves. Arrays of type ARRAY is displayed in bar chart fashion.

The normal curve labeling scheme for ***VPLOT (p. 2146)** is to label curve 1 "COL 1", curve 2 "COL 2" and so on. You can use the **/GCOLUMN (p. 795)** command to apply user-specified labels (8 characters maximum) to your curves. See Modifying Curve Labels in the *Ansys Parametric Design Language Guide* for more information on using **/GCOLUMN (p. 795)**.

When a graph plot reaches minimum or maximum y-axis limits, the program indicates the condition by clipping the graph. The clip appears as a horizontal magenta line. Mechanical APDL calculates y-axis limits automatically; however, you can modify the (YMIN and YMAX) limits via the **/YRANGE (p. 2201)** command.

This command is valid in any processor.

Menu Paths

Utility Menu>Plot>Array Parameters

VPLOT,*NV1*,*NV2*,*NINC*,*DEGEN*,*SCALE*

Displays the selected volumes.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1*, *NV2*, *NINC

Display volumes from *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL (default), *NV2* and *NINC* are ignored and all selected volumes (**VSEL (p. 2163)**) are displayed.

DEGEN

Degeneracy marker:

(blank)

No degeneracy marker is used (default).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEGE

A red star is placed on keypoints at degeneracies (see the *Modeling and Meshing Guide*). Not available if **/FACET (p. 731)**,WIRE is set.

SCALE

Scale factor for the size of the degeneracy-marker star. The scale is the size in window space (-1 to 1 in both directions) (defaults to .075).

Notes

Displays selected volumes. (Only volumes having areas within the selected area set (**ASEL (p. 192)**) will be plotted.) With PowerGraphics on (**/GRAPHICS (p. 866)**,POWER), **VPLOT (p. 2147)** will display only the currently selected areas. This command is also a utility command, valid anywhere. The degree of tessellation used to plot the volumes is set through the **/FACET (p. 731)** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Check Geom>Show Degeneracy>Plot Degen Volus

Main Menu>Preprocessor>Modeling>Operate>Booleans>Show Degeneracy>Plot Degen Volus

Utility Menu>Plot>Specified Entities>Volumes

Utility Menu>Plot>Volumes

VPTN, *NV1*, *NV2*, *NV3*, *NV4*, *NV5*, *NV6*, *NV7*, *NV8*, *NV9*

Partitions volumes.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1, NV2, NV3, . . . , NV9

Numbers of volumes to be operated on. If *NV1* = ALL, *NV2* to *NV9* are ignored and all selected volumes are used. If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1*.

Notes

Partitions volumes. Generates new volumes which encompass the geometry of all the input volumes. The new volumes are defined by the regions of intersection of the input volumes, and by the complementary (non-intersecting) regions. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Partition>Volumes

---

*VPUT,*ParR*,*Entity*,*ENTNUM*,*Item1*,*IT1NUM*,*Item2*,*IT2NUM*,*KLOOP*

Restores array parameter values into the Mechanical APDL database.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the input vector array parameter. See ***SET (p. 1720)** for name restrictions. The parameter must exist as a dimensioned array (***DIM (p. 530)**) with data input.

Entity

Entity keyword. Valid keywords are shown for *Entity* = in the table below.

ENTNUM

The number of the entity (as shown for *ENTNUM*= in the table below).

Item1

The name of a particular item for the given entity. Valid items are as shown in the *Item1* columns of the table below.

IT1NUM

The number (or label) for the specified *Item1* (if any). Valid *IT1NUM* values are as shown in the *IT1NUM* columns of the table below. Some *Item1* labels do not require an *IT1NUM* value.

Item2, IT2NUM

A second set of item labels and numbers to further qualify the item for which data is to be stored. Most items do not require this level of information.

KLOOP

Field to be looped on:

0 or 2 --

Loop on the *ENTNUM* field (default).

3 --

Loop on the *Item1* field.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 --

Loop on the *IT1NUM* field. Successive items are as shown with *IT1NUM*.

5 --

Loop on the *Item2* field.

6 --

Loop on the *IT2NUM* field. Successive items are as shown with *IT2NUM*.

Notes

The ***VPUT (p. 2149)** command is not supported for PowerGraphics displays. Inconsistent results may be obtained if this command is not used in **/GRAPHICS (p. 866)**, FULL.

Plot and print operations entered via the GUI (**Utility Menu> Pltcrtls, Utility Menu> Plot**) incorporate the **AVPRIN (p. 210)** command. This means that the principal and equivalent values are recalculated. If you use ***VPUT (p. 2149)** to put data back into the database, issue the plot commands from the command line to preserve your data.

This operation is basically the inverse of the ***VGET (p. 2118)** operation. Vector items are put directly (without any coordinate system transformation) into the Mechanical APDL database. Items can only replace existing items of the database and not create new items. Degree of freedom results that are replaced in the database are available for all subsequent postprocessing operations. Other results are changed temporarily and are available mainly for the immediately following print and display operations. The vector specification ***VCUM (p. 2080)** does not apply to this command. The valid labels for the location fields (*Entity*, *ENTNUM*, *Item1*, and *IT1NUM*) are listed below. *Item2* and *IT2NUM* are not currently used. Not all items from the ***VGET (p. 2118)** list are allowed on ***VPUT (p. 2149)**, as putting values into some locations could cause the database to be inconsistent.

This command is valid in any processor.

Table 247: *VPUT - POST1 Items

***Entity* = NODE, *ENTNUM* = *n* (node number)**

Item1	IT1NUM	Description
Valid labels for nodal degree of freedom results are:		
U	X, Y, Z	X, Y, or Z structural displacement.
ROT	X, Y, Z	X, Y, or Z structural rotation.
TEMP		Temperature. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use TBOT, TE2, TE3, . . ., TTOP instead of TEMP. Alternative get functions: TEMP(N), TBOT(N), TE2(N), etc.
PRES		Pressure.
VOLT		Electric potential.
MAG		Magnetic scalar potential.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= NODE,** ***ENTNUM*** **=** ***n*** **(node number)**

Item1	IT1NUM	Description
V	X, Y, Z	X, Y, or Z fluid velocity. X, Y, or Z nodal velocity in a transient structural analysis (analysis with **ANTYPE (p. 162)**,TRANS).
A	X, Y, Z	X, Y, or Z magnetic vector potential. X, Y, or Z nodal acceleration in a transient structural analysis (analysis with **ANTYPE (p. 162)**,TRANS).
CONC		Concentration.
CURR		Current.
EMF		Electromotive force drop.

Valid labels for element nodal results are:

Item1	IT1NUM	Description
S	X, Y, Z, XY, YZ, XZ	Component stress.
"	1, 2, 3	Principal stress.
"	INT, EQV	Stress intensity or equivalent stress.
EPTO	X, Y, Z, XY, YZ, XZ	Component total strain (EPEL + EPPL + EPCR).
"	1,2,3	Principal total strain.
"	INT, EQV	Total strain intensity or total equivalent strain.
EPEL	X, Y, Z, XY, YZ, XZ	Component elastic strain.
"	1, 2, 3	Principal elastic strain.
"	INT, EQV	Elastic strain intensity or elastic equivalent strain.
EPPL	X, Y, Z, XY, YZ, XZ	Component plastic strain.
"	1,2,3	Principal plastic strain.
"	INT, EQV	Plastic strain intensity or plastic equivalent strain.
EPCR	X, Y, Z, XY, YZ, XZ	Component creep strain.
"	1, 2, 3	Principal creep strain.
"	INT, EQV	Creep strain intensity or creep equivalent strain.
EPTH	X, Y, Z, XY, YZ, XZ	Component thermal strain.
"	1, 2, 3	Principal thermal strain.
"	INT, EQV	Thermal strain intensity or thermal equivalent strain.
EPSW		Swelling strain.
NL	SEPL	Equivalent stress (from stress-strain curve).
"	SRAT	Stress state ratio.
"	HPRES	Hydrostatic pressure.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity **= NODE,** ***ENTNUM*** **=** ***n*** **(node number)**

Item1	IT1NUM	Description
"	EPEQ	Accumulated equivalent plastic strain.
"	CREQ	Accumulated equivalent creep strain.
"	PSV	Plastic state variable.
"	PLWK	Plastic work/volume.
TG	X, Y, Z	Component thermal gradient.
TF	X, Y, Z	Component thermal flux.
PG	X, Y, Z	Component pressure gradient.
EF	X, Y, Z	Component electric field.
D	X, Y, Z	Component electric flux density.
H	X, Y, Z	Component magnetic field intensity.
B	X, Y, Z	Component magnetic flux density.
FMAG	X, Y, Z	Component electromagnetic force.

Entity **= ELEM,** ***ENTNUM*** **=** ***n*** **(element number)**

Valid labels for element results are:

Item1	IT1NUM	Description
ETAB	Lab	Any user-defined element table label (**ETABLE (p. 687)**).

Menu Paths

Utility Menu>Parameters>Array Operations>Put Array Data

VPUT,*Par*,*IR*,*TSTRT*,*KCPLX*,*Name*

Moves an array parameter vector into a variable.

POST26 (p. 58): Special Purpose (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Par

Array parameter vector in the operation.

IR

Arbitrary reference number assigned to this variable (1 to *NV* (**NUMVAR (p. 1302)**)). Overwrites any existing results for this variable.

TSTRT

Time (or frequency) corresponding to start of *IR* data. If between values, the nearer value is used.

KCPLX

Complex number key:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Use the real part of the *IR* data.

1

Use the imaginary part of the *IR* data.

Name

Thirty-two character name identifying the item on printouts and displays. Defaults to the label formed by concatenating VPUT with the reference number *IR*.

Notes

At least one variable should be defined (**NSOL (p. 1281)**, **ESOL (p. 672)**, **RFORCE (p. 1607)**, etc.) before using this command. The starting array element number must be defined. For example, **VPUT (p. 2152)**,A(1),2 moves array parameter A to variable 2 starting at time 0.0. Looping continues from array element A(1) with the index number incremented by one until the variable is filled. Unfilled variable locations are assigned a zero value. The number of loops may be controlled with the ***VLEN (p. 2136)** command (except that loop skipping (NINC) is not allowed). For multi-dimensioned array parameters, only the first (row) subscript is incremented.

Menu Paths

Main Menu>TimeHist Postpro>Table Operations>Parameter to Var

***VREAD**,*ParR*,*Fname*,*Ext*,--,*Label*,*n1*,*n2*,*n3*,*NSKIP*

Reads data and produces an array parameter vector or matrix.

APDL (p. 19): Parameters (p. 19)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting array parameter vector. See ***SET (p. 1720)** for name restrictions. The parameter must exist as a dimensioned array (***DIM (p. 530)**). String arrays are limited to a maximum of 8 characters.

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

If the *Fname* field is left blank, reading continues from the current input device, such as the terminal.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ext

Filename extension (eight-character maximum).

--

Unused field.

Label

Can take a value of IJK, IKJ, JIK, JKI, KIJ, KJI, or blank (IJK).

n1, n2, n3

Read as (((*ParR* (i,j,k), k = 1,n1), i = 1, n2), j = 1, n3) for *Label* = KIJ. *n2* and *n3* default to 1.

NSKIP

Number of lines at the beginning of the file being read that will be skipped during the reading. Default = 0.

Notes

Reads data from a file and fills in an array parameter vector or matrix. Data are read from a formatted file or, if the menu is off (**/MENU (p. 1121)**,OFF) and *Fname* is blank, from the next input lines. The format of the data to be read must be input immediately following the ***VREAD (p. 2153)** command. The format specifies the number of fields to be read per record, the field width, and the placement of the decimal point (if none specified in the value). The read operation follows the available FORTRAN FORMAT conventions of the system (see your system FORTRAN manual). Any standard FORTRAN *real* format (such as (4F6.0), (E10.3,2X,D8.2), etc.) or alphanumeric format (A) may be used. Alphanumeric strings are limited to a maximum of 8 characters for any field (A8). For storage of string arrays greater than 8 characters, the *SREAD command can be used. Integer (I) and list-directed (*) descriptors may *not* be used. *The parentheses must be included in the format* and the format must not exceed 80 characters (including parentheses). The input line length is limited to 128 characters.

A starting array element number must be defined for the result array parameter vector (numeric or character). For example, entering these two lines:

```
*VREAD,A(1),ARRAYVAL
(2F6.0)
```

will read two values from each line of file ARRAYVAL and assign the values to A(1), A(2), A(3), etc. Reading continues until successive row elements (***VLEN (p. 2136)**, ***VMASK (p. 2139)**, ***DIM (p. 530)**) are filled.

For an array parameter matrix, a starting array element row and column number must be defined. For example, entering these two lines:

```
*VREAD,A(1,1),ARRAYVAL,,,IJK,10,2
(2F6.0)
```

will read two values from each line of file ARRAYVAL and assign the values to A(1,1), A(2,1), A(3,1), etc. Reading continues until *n1* (10) successive row elements are filled. Once the maximum row number is reached, subsequent data will be read into the next column (for example, A(1,2), A(2,2), A(3,2), etc.)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For numerical parameters, absolute values and scale factors may be applied to the result parameter (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). See the ***VOPER (p. 2142)** command for details. If you are in the GUI the ***VREAD (p. 2153)** command must be contained in an externally prepared file read into Mechanical APDL (that is, ***USE (p. 2059)**, **/INPUT (p. 948)**, etc.).

This command is not applicable to 4- or 5-D arrays.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Parameters>Read from File

VROTAT, *NA1*, *NA2*, *NA3*, *NA4*, *NA5*, *NA6*, *PAX1*, *PAX2*, *ARC*, *NSEG*

Generates cylindrical volumes by rotating an area pattern about an axis.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NA1, NA2, NA3, . . . , NA6

List of areas in the pattern to be rotated (6 maximum if using keyboard entry). Areas must lie to one side of, and in the plane of, the axis of rotation. If *NA1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). If *NA1* = ALL, all selected areas will define the pattern to be rotated. A component name may also be substituted for *NA1*.

PAX1, PAX2

Keypoints defining the axis about which the area pattern is to be rotated.

ARC

Arc length (in degrees). Positive follows right-hand rule about *PAX1-PAX2* vector. Defaults to 360.

NSEG

Number of volumes (8 maximum) around circumference. Defaults to minimum required for 90° (maximum) arcs, that is, 4 for 360°, 3 for 270°, etc.

Notes

Generates cylindrical volumes (and their corresponding keypoints, lines, and areas) by rotating an area pattern (and its associated line and keypoint patterns) about an axis. Keypoint patterns are generated at regular angular locations (based on a maximum spacing of 90°). Line patterns are generated at the keypoint patterns. Arc lines are also generated to connect the keypoints circumferentially. Keypoint, line, area, and volume numbers are automatically assigned (beginning with the lowest available values). Adjacent lines use a common keypoint, adjacent areas use a common line, and adjacent volumes use a common area.

To generate a single volume with an arc greater than 180°, *NSEG* must be greater than or equal to 2.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If element attributes have been associated with the input area via the **AATT (p. 85)** command, the opposite area generated by the **VROTAT (p. 2155)** operation will also have those attributes (that is, the element attributes from the input area are copied to the opposite area). Note that only the area opposite the input area will have the same attributes as the input area; the areas adjacent to the input area will not.

If the given areas are meshed or belong to meshed volumes, the 2D mesh can be rotated (extruded) to a 3D mesh. See the *Modeling and Meshing Guide* for more information. Note that the *NDIV* argument on the **ESIZE (p. 668)** command should be set before extruding the meshed areas.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Extrude>Areas>About Axis

---

VSBA, *NV*, *NA*, *SEP0*, *KEEPV*, *KEEPA*

Subtracts areas from volumes.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV

Volume (or volumes, if picking is used) to be subtracted from. If ALL, use all selected volumes. If P, graphical picking is enabled (valid only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NV*.

NA

Area (or areas, if picking is used) to subtract. If ALL, use all selected areas. A component name may also be substituted for *NA*.

SEP0

Behavior of the touching boundary:

(blank)

The resulting volumes will share area(s) where they touch.

SEPO

The resulting volumes will have separate, but coincident area(s) where they touch.

KEEPV

Specifies whether *NV* volumes are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELETE

Delete *NV* volumes after **VSBA (p. 2156)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NV* volumes after **VSBA (p. 2156)** operation (override **BOPTN (p. 269)** command settings).

KEEPA

Specifies whether *NA* areas are to be deleted:

(blank)

Use the setting of KEEP on the **BOPTN (p. 269)** command.

DELETE

Delete *NA* areas after **VSBA (p. 2156)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NA* areas after **VSBA (p. 2156)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new volumes by subtracting the regions common to both the volumes and areas (the intersection) from the *NV* volumes. The intersection will be an area(s). If *SEP0* is blank, the volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEP0* is set to SEPO, the volume is divided into two unconnected volumes with separate areas where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Volume by Area

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Volume by Area

---

VSBV, *NV1*, *NV2*, *SEP0*, *KEEP1*, *KEEP2*

Subtracts volumes from volumes.

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV1

Volume (or volumes, if picking is used) to be subtracted from. If ALL, use all selected volumes. Volumes specified in set *NV2* are removed from set *NV1*. If P, graphical picking is enabled (valid

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

only in the GUI) and remaining fields are ignored. A component name may also be substituted for *NV1*.

NV2

Volume (or volumes, if picking is used) to subtract. If ALL, use all selected volumes (except those included in the *NV1* argument). A component name may also be substituted for *NV2*.

SEP0

Behavior if the intersection of the *NV1* volumes and the *NV2* volumes is an area or areas:

(blank)

The resulting volumes will share area(s) where they touch.

SEPO

The resulting volumes will have separate, but coincident area(s) where they touch.

KEEP1

Specifies whether *NV1* volumes are to be deleted:

(blank)

Use the setting of *KEEP* on the **BOPTN (p. 269)** command.

DELETE

Delete *NV1* volumes after **VSBV (p. 2157)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NV1* volumes after **VSBV (p. 2157)** operation (override **BOPTN (p. 269)** command settings).

KEEP2

Specifies whether *NV2* volumes are to be deleted:

(blank)

Use the setting of *KEEP* on the **BOPTN (p. 269)** command.

DELETE

Delete *NV2* volumes after **VSBV (p. 2157)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NV2* volumes after **VSBV (p. 2157)** operation (override **BOPTN (p. 269)** command settings).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Generates new volumes by subtracting the regions common to both *NV1* and *NV2* volumes (the intersection) from the *NV1* volumes. The intersection can be a volume(s) or area(s). If the intersection is an area and *SEP0* is blank, the *NV1* volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEP0* is set to SEPO, *NV1* is divided into two unconnected volumes with separate areas where they touch. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated. **VSBV (p. 2157)**,ALL,ALL will have no effect because all the volumes in set *NV1*will have been moved to set *NV2*.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>Volumes

Main Menu>Preprocessor>Modeling>Operate>Booleans>Subtract>With Options>Volumes

Main Menu>Preprocessor>Modeling>Operate>Subract>Volumes

VSBW, *NV*, *SEP0*, *KEEP*

Subtracts intersection of the working plane from volumes (divides volumes).

PREP7 (p. 22): Booleans (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NV

Volume (or volumes, if picking is used) to be subtracted from. If *NV* = ALL, use all selected volumes. If *NV* = P, graphical picking is enabled (valid only in the GUI). A component name may also be input for *NV*.

SEP0

Behavior of the created boundary.

(blank)

The resulting volumes will share area(s) where they touch.

SEPO

The resulting volumes will have separate, but coincident area(s).

KEEP

Specifies whether *NV* volumes are to be deleted.

(blank)

Use the setting of *KEEP* on the **BOPTN (p. 269)** command.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DELETE

Delete *NV* volumes after **VSBW (p. 2159)** operation (override **BOPTN (p. 269)** command settings).

KEEP

Keep *NV* volumes after **VSBW (p. 2159)** operation (override **BOPTN (p. 269)** command settings).

Notes

Generates new volumes by subtracting the intersection of the working plane from the *NV* volumes. The intersection will be an area(s). If *SEPO* is blank, the volume is divided at the area and the resulting volumes will be connected, sharing a common area where they touch. If *SEPO* is set to SEPO, the volume is divided into two unconnected volumes with separate areas. The SEPO option may cause unintended consequences if any keypoints exist along the cut plane. See the *Modeling and Meshing Guide* for an illustration. See the **BOPTN (p. 269)** command for an explanation of the options available to Boolean operations. Element attributes and solid model boundary conditions assigned to the original entities will not be transferred to the new entities generated.

Issuing the **VSBW (p. 2159)** command under certain conditions may generate a topological degeneracy error. Do not issue the command if:

- A sphere or cylinder has been scaled. (A cylinder must be scaled unevenly in the XY plane.)
- A sphere or cylinder has not been scaled but the work plane has been rotated.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>Volu by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Booleans>Divide>With Options>Volu by WrkPlane

Main Menu>Preprocessor>Modeling>Operate>Divide>Volu by WrkPlane

---

/VSCALE, *WN*, *VRATIO*, *KEY*

Scales the length of displayed vectors.

GRAPHICS (p. 16): Scaling (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

VRATIO

Ratio value applied to the automatically calculated scale factor (defaults to 1.0, that is, use scale factor as automatically calculated).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KEY

Relative scaling key:

0

Use relative length scaling among vectors based on magnitudes.

1

Use uniform length scaling for all vector lengths.

Command Default

Automatic vector scaling.

Notes

Allows scaling of the vector length displayed with the **PLVECT (p. 1445)** command of POST1 and the **/PBC (p. 1356)** and **/PSF (p. 1537)** commands. Also allows the scaling of the element (that is, **/PSYMB (p. 1552)**,ESYS) and the nodal (that is, **/PSYMB (p. 1552)**,NDIR) coordinate system symbols.

This command is valid in any processor.

Menu Paths

Main Menu>General Postproc>Plot Results>Vector Plot>Predefined

Utility Menu>Plot>Results>Vector Plot

Utility Menu>PlotCtrls>Multi-Plot Contrls

Utility Menu>PlotCtrls>Style>Vector Arrow Scaling

***VSCFUN**,*ParR*,*Func*,*Par1*

Determines properties of an array parameter.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

ParR

The name of the resulting scalar parameter. See ***SET (p. 1720)** for name restrictions.

Func

Functions:

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MAX --

Maximum: the maximum *Par1* array element value.

MIN --

Minimum: the minimum *Par1* array element value.

LMAX --

Index location of the maximum *Par1* array element value. Array *Par1* is searched starting from its specified index.

LMIN --

Index location of the minimum *Par1* array element value. Array *Par1* is searched starting from its specified index.

FIRST --

Index location of the first nonzero value in array *Par1*. Array *Par1* is searched starting from its specified index.

LAST --

Index location of the last nonzero value in array *Par1*. Array *Par1* is searched starting from its specified index.

SUM --

Sum: *Par1* (the summation of the *Par1* array element values).

MEDI --

Median: value of *Par1* at which there are an equal number of values above and below.

MEAN --

Mean: (σ Par1)/NUM, where NUM is the number of summed values.

VARI --

Variance: (σ ((*Par1*-MEAN)**2))/NUM.

STDV --

Standard deviation: square root of VARI.

RMS --

Root-mean-square: square root of (σ (*Par1***2))/NUM.

NUM --

Number: the number of summed values (masked values are not counted).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Par1

Array parameter vector in the operation.

Notes

Operates on one input array parameter vector and produces one output scalar parameter according to:

ParR = f(*Par1*)

where the functions (f) are described below. The starting array element number must be defined for the array parameter vector. For example, ***VSCFUN (p. 2161)**,MU,MEAN,A(1) finds the mean of the A vector values, starting from the first value and stores the result as parameter MU. Operations use successive array elements (***VLEN (p. 2136)**, ***VMASK (p. 2139)**) with the default being all successive array elements. Absolute values and scale factors may be applied to all parameters (***VABS (p. 2072)**, ***VFACT (p. 2092)**). Results may be cumulative (***VCUM (p. 2080)**). See the ***VOPER (p. 2142)** command for details.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Operations>Vector-Scalar Func

VSEL, *Type*, *Item*, *Comp*, *VMIN*, *VMAX*, *VINC*, *KSWP*

Selects a subset of volumes.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of volume select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ALL

Restore the full set.

NONE

Unselect the full set.

INVE

Invert the current set (selected becomes unselected and vice versa).

STAT

Display the current select status.

The following fields are used only with *Type* = S, R, A, or U:

Item

Label identifying data. Valid item labels are shown in the table below. Some items also require a component label. If *Item* = PICK (or simply "P"), graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). Defaults to VOLU.

Comp

Component of the item (if required). Valid component labels are shown in the table below.

VMIN

Minimum value of item range. Ranges are volume numbers, coordinate values, attribute numbers, etc., as appropriate for the item. A component name (as specified on the **CM (p. 356)** command) may also be substituted for *VMIN* (*VMAX* and *VINC* are ignored). If *Item* = MAT, TYPE, REAL, or ESYS and if *VMIN* is positive, the absolute value of *Item* is compared against the range for selection; if *VMIN* is negative, the signed value of *Item* is compared. See the **VLIST (p. 2137)** command for a discussion of signed attributes.

VMAX

Maximum value of item range. *VMAX* defaults to *VMIN*.

VINC

Value increment within range. Used only with integer ranges (such as for volume numbers). Defaults to 1. *VINC* cannot be negative.

KSWP

Specifies whether only volumes are to be selected:

0

Select volumes only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Select volumes, as well as keypoints, lines, areas, nodes, and elements associated with selected volumes. Valid only with *Type* = S.

Command Default

All volumes are selected.

Notes

Selects volumes based on values of a labeled item and component. For example, to select a new set of volumes based on volume numbers 1 through 7, use **VSEL (p. 2163)**,S,VOLU,,1,7. The subset is used when the ALL label is entered (or implied) on other commands, such as **VLIST (p. 2137)**,ALL. Only data identified by volume number are selected. Data are flagged as selected and unselected; no data are actually deleted from the database.

This command is valid in any processor.

For Selects based on non-integer numbers (coordinates, results, etc.), items that are within the range VMIN-*Toler* and VMAX+*Toler* are selected. The default tolerance *Toler* is based on the relative values of VMIN and VMAX as follows:

- If VMIN = VMAX, *Toler* = 0.005 x VMIN.
- If VMIN = VMAX = 0.0, *Toler* = 1.0E-6.
- If VMAX ≠ VMIN, *Toler* = 1.0E-8 x (VMAX-VMIN).

Use the **SELTOL (p. 1710)** (p. 1710) command to override this default and specify *Toler* explicitly.

Table 248: VSEL (p. 2163) - Valid Item and Component Labels

VSEL (p. 2163) *Type, Item, Comp, VMIN, VMAX, VINC, KABS*

Item	Comp	Description
VOLU		Volume number.
LOC	X, Y, Z	X, Y, or Z center (picking "hot spot" location in the active coordinate system).
MAT		Material number associated with the volume.
TYPE		Element type number associated with the volume.
REAL		Real constant set number associated with the volume.
ESYS		Element coordinate system associated with the volume.

Menu Paths

Utility Menu>Select>Entities

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VSLA, *Type*, *VLKEY*

Selects those volumes containing the selected areas.

DATABASE (p. 13): Selecting (p. 14)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Type

Label identifying the type of volume select:

S

Select a new set (default).

R

Reselect a set from the current set.

A

Additionally select a set and extend the current set.

U

Unselect a set from the current set.

VLKEY

Specifies whether all contained volume areas must be selected (**ASEL (p. 192)**):

0

Select volume if any of its areas are in the selected area set.

1

Select volume only if all of its areas are in the selected area set.

Notes

This command is valid in any processor.

Menu Paths

Utility Menu>Select>Entities

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*VSTAT

Lists the current specifications for the array parameters.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

Lists the current specifications for the ***VABS (p. 2072)**, ***VCOL (p. 2077)**, ***VCUM (p. 2080)**, ***VFACT (p. 2092)**, ***VLEN (p. 2136)**, and ***VMASK (p. 2139)** commands.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

VSUM, *LAB*

Calculates and prints geometry statistics of the selected volumes.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LAB

Controls the degree of tessellation used in the calculation of area properties. If *LAB* = DEFAULT, area calculations will use the degree of tessellation set through the **/FACET (p. 731)** command. If *LAB* = FINE, area calculations are based on a finer tessellation.

Notes

Calculates and prints geometry statistics (volume, centroid location, moments of inertia, etc.) associated with the selected volumes. Geometry items are reported in the global Cartesian coordinate system. A unit density is assumed unless the volumes have a material association via the **VATT (p. 2075)** command. Items calculated by **VSUM (p. 2167)** and later retrieved by a ***GET (p. 797)** or ***VGET (p. 2118)** command are valid only if the model is not modified after the **VSUM (p. 2167)** command is issued.

Setting a finer degree of tessellation will provide area calculations with greater accuracy, especially for thin, hollow models. However, using a finer degree of tessellation requires longer processing.

For very thin volumes, such that the ratio of the minimum to the maximum dimension is less than 0.01, the **VSUM (p. 2167)** command can provide erroneous volume information. To ensure that such calculations are accurate, make certain that you subdivide such volumes so that the ratio of the minimum to the maximum is at least 0.05.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Operate>Calc Geom Items>Of Volumes

---

VSWEEP, *VNUM*, *SRCA*, *TRGA*, *LSMO*

Fills an existing unmeshed volume with elements by sweeping the mesh from an adjacent area through the volume.

PREP7 (p. 22): Meshing (p. 29)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

VNUM

Number identifying the volume that is to be meshed by **VSWEEP (p. 2168)**. If *VNUM* = P, graphical picking is enabled, you will be prompted to choose the volume or volumes based on the setting of **EXTOPT (p. 722)**,VSWE,AUTO. This argument is required.

"ALL" is a valid input value that when selected sends all the selected volumes to the sweeper. If *VNUM* = ALL, each volume that can be swept will be and those not able to be swept will be unmeshed or meshed with tets depending upon the setting of **EXTOPT (p. 722)**,VSWE,TETS.

A component name is a valid input value. All volumes that are part of the specified component will be sent to the sweeper.

SRCA

Number identifying the source area. This is the area whose mesh will provide the pattern for the volume elements. (If you do not mesh the source area prior to volume sweeping, the program meshes it internally when you initiate volume sweeping.) The program sweeps the pattern of the area elements through the volume to create the mesh of volume elements. You cannot substitute a component name for *SRCA*.

This argument is optional. If *VNUM* = ALL or is a component containing more than one volume, *SRCA* is ignored. If *SRCA* is not provided or if it is ignored, **VSWEEP (p. 2168)** attempts to automatically determine which area should be the target area.

TRGA

Number identifying the target area. This is the area that is opposite the source area specified by *SRCA*. You cannot substitute a component name for *TRGA*.

This argument is optional. If *VNUM* = ALL or component containing more than one volume, *TRGA* is ignored. If *TRGA* is not provided or if it is ignored, **VSWEEP (p. 2168)** attempts to automatically determine which area should be the target area.

LSMO

Value specifying whether the program should perform line smoothing during volume sweeping. (The value of this argument controls line smoothing for the **VSWEEP (p. 2168)** command only; it has no effect on the setting of the **MOPT (p. 1151)** command's LSMO option.) This argument is optional.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0

Do not perform line smoothing. This is the default.

1

Always perform line smoothing. This setting is not recommended for large models due to speed considerations.

Notes

If the source mesh consists of quadrilateral elements, the program fills the volume with hexahedral elements. If the source mesh consists of triangles, the program fills the volume with wedges. If the source mesh consists of a combination of quadrilaterals and triangles, the program fills the volume with a combination of hexahedral and wedge elements.

In the past, you may have used the **VROTAT (p. 2155)**, **VEXT (p. 2091)**, **VOFFST (p. 2140)**, and/or **VDRAG (p. 2084)** commands to extrude a meshed area into a meshed volume. However, those commands create the volume and the volume mesh simultaneously. In contrast, the **VSWEEP (p. 2168)** command is intended for use in an existing unmeshed volume. This makes **VSWEEP (p. 2168)** particularly useful when you have imported a solid model that was created in another program, and you want to mesh it in Mechanical APDL.

For related information, see the description of the **EXTOPT (p. 722)** command (although **EXTOPT (p. 722)** sets volume sweeping options, it does not affect element spacing). Also see the detailed discussion of volume sweeping in Meshing Your Solid Model of the *Modeling and Meshing Guide*.

Menu Paths

Main Menu>Preprocessor>Meshing>Mesh>Volume Sweep>Sweep

VSYMM,*Ncomp*,*NV1*,*NV2*,*NINC*,*KINC*,*NOELEM*,*IMOVE*

Generates volumes from a volume pattern by symmetry reflection.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ncomp

Symmetry key:

X

X symmetry (default).

Y

Y symmetry.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z

Z symmetry.

NV1, NV2, NINC

Reflect volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes (**VSEL (p. 2163)**). If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether nodes and elements are also to be generated:

0

Generate nodes and elements associated with the original volumes, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether volumes will be moved or newly defined:

0

Generate additional volumes.

1

Move original volumes to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

Notes

Generates a reflected set of volumes (and their corresponding keypoints, lines, areas and mesh) from a given volume pattern by a symmetry reflection (see analogous node symmetry command, **NSYM (p. 1289)**). The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings. Reflection is done in the active coordinate system by changing a particular coordinate sign. The active coordinate system must be a Cartesian system. Volumes in the pattern may have been generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Volumes are generated as described in the **VGEN (p. 2116)** command.

See the **ESYM (p. 684)** command for additional information about symmetry elements.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Preprocessor>Modeling>Reflect>Volumes

VTRAN, *KCNTO*, *NV1*, *NV2*, *NINC*, *KINC*, *NOELEM*, *IMOVE*

Transfers a pattern of volumes to another coordinate system.

PREP7 (p. 22): Volumes (p. 28)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

KCNTO

Reference number of coordinate system where the pattern is to be transferred. Transfer occurs from the active coordinate system. The coordinate system type and parameters of *KCNTO* must be the same as the active system.

NV1*, *NV2*, *NINC

Transfer volumes from pattern beginning with *NV1* to *NV2* (defaults to *NV1*) in steps of *NINC* (defaults to 1). If *NV1* = ALL, *NV2* and *NINC* are ignored and the pattern is all selected volumes (**VSEL (p. 2163)**). If *NV1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI). A component name may also be substituted for *NV1* (*NV2* and *NINC* are ignored).

KINC

Keypoint increment between sets. If zero, the lowest available keypoint numbers are assigned (**NUMSTR (p. 1301)**).

NOELEM

Specifies whether elements and nodes are also to be generated:

0

Generate nodes and elements associated with the original volumes, if they exist.

1

Do not generate nodes and elements.

IMOVE

Specifies whether to redefine the existing volumes:

0

Generate additional volumes.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1

Move original volumes to new position retaining the same keypoint numbers (*KINC* and *NOELEM* are ignored). Corresponding meshed items are also moved if not needed at their original position.

Notes

Transfers a pattern of volumes (and their corresponding keypoints, lines, areas and mesh) from one coordinate system to another (see analogous node transfer command, **TRANSFER (p. 2011)**). The MAT, TYPE, REAL, and ESYS attributes are based upon the volumes in the pattern and not upon the current settings. Coordinate systems may be translated and rotated relative to each other. Initial pattern may be generated in any coordinate system. However, solid modeling in a toroidal coordinate system is not recommended. Coordinate and slope values are interpreted in the active coordinate system and are transferred directly. Volumes are generated as described in the **VGEN (p. 2116)** command.

Menu Paths

Main Menu>Preprocessor>Modeling>Move / Modify>Transfer Coord>Volumes

---

VTYPE, *NOHID*, *NZONE*

Specifies the viewing procedure used to determine the form factors for the Radiation Matrix method.

AUX12 (p. 62): Radiation Matrix Method (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

NOHID

Type of viewing procedure:

0

Hidden procedure.

1

Non-hidden (faster, but less general) procedure.

NZONE

Number of sampling zones for the hidden procedure (100 maximum for 3D, 1000 maximum for 2D). Defaults to 20 for 3D, 200 for 2D. Number of points is 2**NZONE* for 2D and 2**NZONE**(*NZONE*+1) for 3D.

Command Default

Hidden procedure using 20 zones for 3D, 200 zones for 2D.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Write Matrix

/VUP, *WN*, *Label*

Specifies the global Cartesian coordinate system reference orientation.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number (or ALL) to which command applies (defaults to 1).

Label

Orientation:

Y

Y vertical upward, X horizontal to the right, Z out from the screen (default).

-Y

Y vertical downward, X horizontal to the left, Z out from the screen.

X

X vertical upward, Y horizontal to the left, Z out from the screen.

-X

X vertical downward, Y horizontal to the right, Z out from the screen.

Z

Z vertical upward, Y horizontal to the right, X out from the screen. With this choice, you should use a view other than the **/VIEW (p. 2131)** default of (0,0,1).

-Z

Z vertical downward, Y horizontal to the left, X out from the screen. With this choice, you should use a view other than the **/VIEW (p. 2131)** default of (0,0,1).

Command Default

Y vertical upward, X horizontal to the right, Z out from the screen.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Specifies the global Cartesian coordinate system reference orientation. The **/VIEW (p. 2131)** and **/ANGLE (p. 140)** commands may be used to reorient the view and are relative to this reference orientation. All coordinate systems are right-handed.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>View Settings>Viewing Direction

***VWRITE**,*Par1*,*Par2*,*Par3*,*Par4*,*Par5*,*Par6*,*Par7*,*Par8*,*Par9*,*Par10*,*Par11*,*Par12*,*Par13*,*Par14*,*Par15*,*Par16*,*Par17*,*Par18*,*Par19*

Writes data to a file in a formatted sequence.

APDL (p. 19): Array Parameters (p. 20)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Par1*, *Par2*, *Par3*, . . . , *Par19

You can write up to 19 parameters (or constants) at a time. Any *Par* values after a blank *Par* value are ignored. If you leave them all blank, one line will be written (to write a title or a blank line). If you input the keyword SEQU, a sequence of numbers (starting from 1) will be written for that item.

Notes

You use ***VWRITE (p. 2174)** to write data to a file in a formatted sequence. Data items (*Par1*, *Par2*, etc.) may be array parameters, scalar parameters, character parameters (scalar or array), or constants. You must evaluate expressions and functions in the data item fields before using the ***VWRITE (p. 2174)** command, since initially they will be evaluated to a constant and remain constant throughout the operation. Unless a file is defined with the ***CFOPEN (p. 316)** command, data is written to the standard output file. Data written to the standard output file may be diverted to a different file by first switching the current output file with the **/OUTPUT (p. 1334)** command. You can also use the ***MWRITE (p. 1202)** command to write data to a specified file. Both commands contain format descriptors on the line immediately following the command. The format descriptors can be in either FORTRAN or C format.

You must enclose FORTRAN format descriptors in parentheses. They must immediately follow the ***VWRITE (p. 2174)** command on a separate line of the same input file. Do not include the word FORMAT. The format must specify the number of fields to be written per line, the field width, the placement of the decimal point, etc. You should use one field descriptor for each data item written. The write operation uses your system's available FORTRAN FORMAT conventions (see your system FORTRAN manual). You can use any standard FORTRAN *real* format (such as (4F6.0), (E10.3,2X,D8.2), etc.) and alphanumeric format (A). Alphanumeric strings are limited to a maximum of 8 characters for any field (A8) using the FORTRAN format. Use the "C" format for string arrays larger than 8 characters. Integer (I) and list-directed (*) descriptors may *not* be used. You can include text in the format as a quoted string. *The parentheses*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

must be included in the format and the format must not exceed 80 characters (including parentheses). The output line length is limited to 128 characters.

The "C" format descriptors are used if the first character of the format descriptor line is not a left parenthesis. "C" format descriptors are up to 80 characters long, consisting of text strings and predefined "data descriptors" between the strings where numeric or alphanumeric character data will be inserted. The normal descriptors are %I for integer data, %G for double precision data, %C for alphanumeric character data, and %/ for a line break. There must be one data descriptor for each specified value (8 maximum) in the order of the specified values. The enhanced formats described in ***MSG (p. 1185)** may also be used.

For array parameter items, you must define the starting array element number. Looping continues (incrementing the vector index number of each array parameter by one) each time you output a line, until the maximum array vector element is written. For example, ***VWRITE (p. 2174)**,A(1) followed by (F6.0) will write one value per output line, that is, A(1), A(2), A(3), A(4), etc. You write constants and scalar parameters with the same values for each loop. You can also control the number of loops and loop skipping with the ***VLEN (p. 2136)** and ***VMASK (p. 2139)** commands. The vector specifications ***VABS (p. 2072)**, ***VFACT (p. 2092)**, and ***VCUM (p. 2080)** do not apply to this command. If looping continues beyond the supplied data array's length, zeros will be output for numeric array parameters and blanks for character array parameters. For multi-dimensioned array parameters, only the first (row) subscript is incremented. See the ***VOPER (p. 2142)** command for details. If you are in the GUI, the ***VWRITE (p. 2174)** command must be contained in an externally prepared file and read into Mechanical APDL (that is, ***USE (p. 2059)**, **/INPUT (p. 948)**, etc.).

If *Par* is a table array name, the subscripts refer to the index numbers of the table and not the index values of its primary variables. See ***SET (p. 1720)** if you want to evaluate a table array at certain values of its primary variables for writing.

This command is valid in any processor.

Menu Paths

Utility Menu>Parameters>Array Parameters>Write to File

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

W Commands

/WAIT, *DTIME*

Causes a delay before the reading of the next command.

APDL (p. 19): Process Controls (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

DTIME

Time delay (in seconds). Maximum time delay is 59 seconds.

Notes

The command following the **/WAIT (p. 2177)** will not be processed until the specified wait time increment has elapsed. Useful when reading from a prepared input file to cause a pause, for example, after a display command so that the display can be reviewed for a period of time. Another "wait" feature is available via the ***ASK (p. 84)** command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

/WINDOW, *WN*, *XMIN*, *XMAX*, *YMIN*, *YMAX*, *NCOPY*

Defines the window size on the screen.

GRAPHICS (p. 16): Set Up (p. 16)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window reference number (1 to 5). Defaults to 1. This number, or ALL (for all active windows), may be used on other commands.

XMIN, XMAX, YMIN, YMAX

Screen coordinates defining window size. Screen coordinates are measured as -1.0 to 1.67 with the origin at the screen center. For example, (-1,1.67,-1,1) is full screen, (-1,0,-1,0) is the left bottom

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

quadrant. If *XMIN* = OFF, deactivate this previously defined window; if ON, reactivate this previously defined window. If FULL, LEFT, RIGH, TOP, BOT, LTOP, LBOT, RTOP, RBOT, form full, half, or quarter window. If SQUA, form largest square window within the current graphics area. If DELE, delete this window (cannot be reactivated with ON).

NCOPY

Copies the current specifications from window *NCOPY* (1 to 5) to this window. If *NCOPY* = 0 (or blank), no specifications are copied.

Command Default

One window at full screen.

Notes

Defines the window size on the screen. Windows may occupy a separate section of the screen or they may overlap. Requested displays are formed in all windows according to the selected window specifications.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Window Controls>Copy Window Specs

Utility Menu>PlotCtrls>Window Controls>Delete Window

Utility Menu>PlotCtrls>Window Controls>Window Layout

Utility Menu>PlotCtrls>Window Controls>Window On or Off

WPAVE,*X1*,*Y1*,*Z1*,*X2*,*Y2*,*Z2*,*X3*,*Y3*,*Z3*

Moves the working plane origin to the average of specified points.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X1, Y1, Z1

Coordinates (in the active coordinate system) of the first point. If *X1* = P, graphical picking is enabled and all remaining command fields are ignored (valid only in the GUI).

X2, Y2, Z2

Coordinates (in the active coordinate system) of the second point.

X3, Y3, Z3

Coordinates (in the active coordinate system) of the third point.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Notes

Moves the origin of the working plane to the average of the specified points. A point is considered specified only if at least one of its coordinates is non-blank, and at least one point (1, 2, or 3) must be specified. Blank coordinates of a specified point are assumed to be zero. Averaging is based on the active coordinate system.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP to>Global Origin

Utility Menu>WorkPlane>Offset WP to>Origin of Active CS

Utility Menu>WorkPlane>Offset WP to>XYZ Locations

WPCSYS, *WN*, *KCN*

Defines the working plane location based on a coordinate system.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified.

KCN

Coordinate system number. *KCN* may be 0,1,2 or any previously defined local coordinate system number (defaults to the active system).

Notes

Defines a working plane location and orientation based on an existing coordinate system. If a Cartesian system is used as the basis (*KCN*) for the working plane, the working plane will also be Cartesian, in the X-Y plane of the base system. If a cylindrical, spherical, or toroidal base system is used, the working plane will be a polar system in the R-θ plane of the base system.

If working plane tracking has been activated (**CSYS (p. 441)**,WP or **CSYS (p. 441)**,4), the updated active coordinate system will be of a similar type, except that a toroidal system will be updated to a cylindrical system. See the *Modeling and Meshing Guide* for more information on working plane tracking.

This command is valid in any processor.

Some primitive generation commands will not honor R-theta transformations for non-cartesian coordinate systems. Refer to the primitive commands table (p. 25) for more information.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Main Menu>General Postproc>Surface Operations>Create Surface>Sphere>At Node

Utility Menu>WorkPlane>Align WP with>Active Coord Sys

Utility Menu>WorkPlane>Align WP with>Global Cartesian

Utility Menu>WorkPlane>Align WP with>Specified Coord Sys

WPLANE, *WN*, *XORIG*, *YORIG*, *ZORIG*, *XXAX*, *YXAX*, *ZXAX*, *XPLAN*, *YPLAN*, *ZPLAN*

Defines a working plane to assist in picking operations.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number whose viewing direction will be modified to be normal to the working plane (defaults to 1). If *WN* is a negative value, the viewing direction will not be modified. If fewer than three points are used, the viewing direction of window *WN* will be used instead to define the normal to the working plane.

XORIG*, *YORIG*, *ZORIG

Global Cartesian coordinates of the origin of the working plane coordinate system.

XXAX*, *YXAX*, *ZXAX

Global Cartesian coordinates of a point defining the x-axis orientation. The x-axis aligns with the projection of the line from this orientation point to the origin.

XPLAN*, *YPLAN*, *ZPLAN

Global Cartesian coordinates of the third point defining the working plane. This point will also define the location of the positive XY-sector of the working plane coordinate system.

Command Default

Working plane parallel to the global X-Y plane at Z = 0.0.

Notes

Defines a working plane to assist in picking operations using the coordinates of three noncolinear points. The three points also define the working plane coordinate system. A minimum of one point (the working plane origin) is required. Immediate mode may also be active. See **WPSTYL (p. 2182)** command to set the style of working plane display.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>WorkPlane>Align WP with>XYZ Locations

WPOFFS, *X0FF*, *Y0FF*, *Z0FF*

Offsets the working plane.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

X0FF*, *Y0FF*, *Z0FF

Offset increments defined in the working plane coordinate system. If only *Z0FF* is used, the working plane will be redefined parallel to the present plane and offset by *Z0FF*.

Notes

Changes the origin of the working plane by translating the working plane along its coordinate system axes.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP by Increments

WPROTA, *THXY*, *THYZ*, *THZX*

Rotates the working plane.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

THXY

First rotation about the working plane Z axis (positive X toward Y).

THYZ

Second rotation about working plane X axis (positive Y toward Z).

THZX

Third rotation about working plane Y axis (positive Z toward X).

Notes

The specified angles (in degrees) are relative to the orientation of the working plane.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command is valid in any processor.

Menu Paths

Utility Menu>WorkPlane>Offset WP by Increments

WPSTYL,*SNAP*,*GRSPAC*,*GRMIN*,*GRMAX*,*WPTOL*,*WPCTYP*,*GRTYPE*,*WPVIS*,*SNAPANG*

Controls the display and style of the working plane.

DATABASE (p. 13): Working Plane (p. 15)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

SNAP

Snap increment for a locational pick (1E-6 minimum). If -1, turn off snap capability. For example, a picked location of 1.2456 with a snap of 0.1 gives 1.2, with 0.01 gives 1.25, with 0.001 gives 1.246, and with 0.025 gives 1.250 (defaults to 0.05).

GRSPAC

Graphical spacing between grid points. For graphical representation only and not related to snap points (defaults to 0.1).

GRMIN*, *GRMAX

Defines the size of a square grid (if *WPCTYP* = 0) to be displayed over a portion of the working plane. The opposite corners of the grid will be located at grid points nearest the working plane coordinates of (*GRMIN*,*GRMIN*) and (*GRMAX*,*GRMAX*). If a polar system (*WPCTYP* = 1), *GRMAX* is the outside radius of grid and *GRMIN* is ignored. If *GRMIN* = *GRMAX*, no grid will be displayed (defaults to -1.0 and 1.0 for *GRMIN* and *GRMAX* respectively).

WPTOL

The tolerance that an entity's location can deviate from the specified working plane, while still being considered on the plane. Used only for locational picking of vertices for polygons and prisms (defaults to 0.003).

WPCTYP

Working plane coordinate system type:

0

Cartesian (default). If working plane tracking is on (**CSYS (p. 441)**,4), the updated active coordinate system will also be Cartesian.

1

Polar. If working plane tracking is on, the updated active coordinate system will be cylindrical.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2

Polar. If working plane tracking is on, the updated active coordinate system will be spherical.

GRTYPE

Grid type:

0

Grid and WP triad.

1

Grid only.

2

WP triad only (default).

WPVIS

Grid visibility:

0

Do not show *GRTYPE* entities (grid and/or triad) (default).

1

Show *GRTYPE* entities. Cartesian working planes will be displayed with a Cartesian grid, polar with a polar grid.

SNAPANG

Snap angle (0--180) in degrees. Used only if *WPCTYP* = 1 or 2. Defaults to 5 degrees.

Command Default

Snap capability on with an increment of .05, spacing between grid points of 0.1, Cartesian coordinate system, WP triad displayed, and a tolerance of .003.

Notes

Use **WPSTYL (p. 2182)**,DEFA to reset the working plane to its default location and style. Use **WPSTYL (p. 2182)**,STAT to list the status of the working plane. Blank fields will keep present settings.

It is possible to specify *SNAP* and *WPTOL* values that will cause conflicts during picking operations. Check your values carefully, and if problems are noted, revert to the default values.

WPSTYL (p. 2182) with no arguments will toggle the grid on and off. The working plane can be displayed in the non-GUI interactive mode only after issuing a **/PLOPTS (p. 1435)**,WP,1 command. See the *Modeling*

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and Meshing Guide for more information on working plane tracking. See **/PLOPTS (p. 1435)** command for control of hidden line working plane.

This command is valid in any processor.

Menu Paths

Main Menu>Preprocessor>Modeling>Create>Circuit>Set Grid

Utility Menu>List>Status>Working Plane

Utility Menu>WorkPlane>Display Working Plane

Utility Menu>WorkPlane>Offset WP by Increments

Utility Menu>WorkPlane>Show WP Status

Utility Menu>WorkPlane>WP settings

WRFULL, *Ldstep*

Stops solution after assembling global matrices.

SOLUTION (p. 38): Load Step Options (p. 43)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Ldstep

Specify action to take:

OFF or 0

Turn off feature (default)

N

Turn on feature and set it to stop after assembling the global matrices and writing the `.full` file for load step N.

Command Default

By default the WRFULL command is turned OFF.

Notes

This command is used in conjunction with the **SOLVE (p. 1822)** command to generate the assembled matrix file (`.full` file) only. The element matrices are assembled into the relevant global matrices for the particular analysis being performed and the `.full` file is written. Equation solution and the output of data to the results file are skipped. To dump the matrices written on the `.full` file into Harwell-Boeing format, use the **HBMAT (p. 888)** command in /AUX2. To copy the matrices to a postscript format that can be viewed graphically, use the **PSMAT (p. 1540)** command.

To use the **LSSOLVE (p. 1085)** macro with this command, you may need to modify the **LSSOLVE (p. 1085)** macro to properly stop at the load step of interest.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command only valid for linear static, full harmonic, and full transient analyses when the sparse direct solver is selected. This command is also valid for buckling or modal analyses with any mode extraction method. This command is not valid for nonlinear analyses. It is not supported in a linear perturbation analysis.

In general, the assembled matrix file `.full` contains stiffness, mass, and damping matrices. However, the availability of the matrices depends on the analysis type chosen when the file is written. Some analyses do not write the matrices individually but instead write combined matrices. For example, a full transient writes a combined stiffness/mass/damping matrix to the full file.

Menu Paths

Main Menu>Preprocessor>Loads>Load Step Opts>Stop Solution

Main Menu>Solution>Load Step Opts>Stop Solution

WRITE, *Fname*

Writes the radiation matrix file.

AUX12 (p. 62): Radiation Matrix Method (p. 62)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

The file name Defaults to `Jobname`.

Notes

Writes radiation matrix file (`File.sub`) for input to the substructure thermal "use" pass. Subsequent **WRITE (p. 2185)** operations to the same file overwrite the file.

Menu Paths

Main Menu>Radiation Opt>Matrix Method>Write Matrix

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WRITEMAP, *Fname*

Writes interpolated pressure data to a file.

/MAP (p. 63): Pressure Mapping (p. 63)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Fname

File name and directory path (248 characters maximum, including the characters needed for the directory path). An unspecified directory path defaults to the working directory; in this case, you can use all 248 characters for the file name.

Notes

Writes the interpolated pressure data to the specified file. The data is written as **SFE (p. 1761)** commands applied to the SURF154 elements that are on the target surface. You may read this data for inclusion in an analysis by using **/INPUT (p. 948)**,*Fname*.

Menu Paths

This command cannot be accessed from a menu.

***WRK**, *Num*

Sets the active workspace number.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Argument Descriptions

Num

Number of the active memory workspace for APDLMath vector and matrices. All the following APDLMath vectors and matrices will belong to this memory workspace, until the next call to the ***WRK (p. 2186)** command. By default, all the APDLMath objects belong to workspace number 1.

Notes

This feature enables you to associate a set of vector and matrices in a given memory workspace, so that you can easily manage the free step:

```
*VEC,V,D,ALLOC,5  ! V belongs to the default Workspace 1

*WRK,2     ! Set the active workspace as the number 2

*VEC,W,D,IMPORT,FULL,file.full,RHS ! W belongs to the Workspace 2
*SMAT,K,D,IMPORT,FULL,file.full,STIFF ! K belongs to the Workspace 2
*DMAT,M,ALLOC,10,10    ! M belongs to the Workspace 2
…
*FREE,WRK,2   ! W, K and M are deleted, but not V
```


Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
*PRINT,V
```

This feature can be useful to free all the temporary APDLMath variables inside a MACRO in one call.

Menu Paths

This command cannot be accessed from a menu.

WSPRINGS

Creates weak springs on corner nodes of a bounding box of the currently selected elements.

SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Notes

This command invokes a predefined Mechanical APDL macro that is used during the import of loads from the ADAMS program into Mechanical APDL.

The command creates weak springs on the corner nodes of the bounding box of the currently selected elements. The six nodes of the bounding box are attached to ground using COMBIN14 elements. The stiffness is chosen as a small number and can be changed by changing the real constants of the COMBIN14 elements.

The command works only for models that have a geometric extension in two or three dimensions. One-dimensional problems (pure beam in one axis) are not supported.

For more information about how **WSPRINGS (p. 2187)** is used during the transfer of loads from the ADAMS program to Mechanical APDL, see Import Loads into Mechanical APDL.

Distributed-Memory Parallel (DMP) Restriction

This command is not supported in a DMP solution.

Menu Paths

Main Menu>Solution>ADAMS Connection>Import fr ADAMS

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WTBCREATE, *IEL*, *NODE*, *DAMP*

Creates a USER300 element to model the turbine for full aeroelastic coupling analysis and specifies relevant settings for the analysis.

PREP7 (p. 22): Elements (p. 32)

Valid Products: (p. 2) Pro | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

IEL

Element number (next available number by default).

NODE

Node number connecting support structure and turbine.

DAMP

Damping option for the turbine:

0

Damping matrix obtained from the aeroelastic code plus Rayleigh damping (default).

1

Rayleigh damping only.

2

Damping from the aeroelastic code only.

Notes

WTBCREATE (p. 2188) invokes a predefined Mechanical APDL macro that automatically generates a turbine element and issue relevant data commands that are necessary to run a full aeroelastic coupling analysis. For detailed information on how to perform a fully coupled aeroelastic analysis, see Fully Coupled Wind Turbine Example in Mechanical APDL in the *Programmer's Reference*.

The generated USER300 turbine element will have 9 nodes with node numbers NODE, NMAX+1, NMAX+2, ..., NMAX+8, where NMAX is the maximum node number currently in the model.

There are six freedoms on the first node of the element: UX, UY, UZ, ROTX, ROTY, ROTZ, and these are true structural freedoms. For all the other nodes (that is, nodes 2 to 9), only the translational freedoms (UX, UY, UZ) are used. These are generalized freedoms that are internal to the turbine element and are used by the aeroelastic code only.

The element type integer of the USER300 element is the current maximum element type integer plus one.

The command will also set up the analysis settings appropriate for a full aeroelastic coupling analysis. These include full Newton-Raphson solution (**NROPT (p. 1261)**,FULL) and a **USRCAL (p. 2062)** command to activate the relevant user routines.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

X Commands

XFCRKMESH,*EnrichmentID*,*ElemComp*,*NodeComp*

Defines a crack in the model when the crack surface is discretized by MESH200 elements

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

EnrichmentID

Name of the enrichment specified via the associated **XFENRICH (p. 2193)** command.

ElemComp

Name of the element component consisting of MESH200 elements that form the crack surface.

NodeComp

Name of the node component consisting of the crack front nodes of the crack surface.

Notes

Used in an XFEM-based crack analysis, this command defines a crack in the model when the crack surface is discretized by MESH200 elements. For more informatiom, see MESH200 Element Method in the *Fracture Analysis Guide*.

Issue the **XFCRKMESH (p. 2191)** command multiple times as needed to define multiple crack surfaces in the model.

This command is valid in PREP7 (**/PREP7 (p. 1479)**) only.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

XFDATA, *EnrichmentID*, LSM or --, *ELEMNUM*, *NODENUM*, *PHI*, *PSI*

Defines a crack in the model by specifying nodal level set values

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

EnrichmentID

Name of the enrichment specified via the associated **XFENRICH (p. 2193)** command.

LSM

Indicates that level set values are being specified (default).

ELEMNUM

Element number.

NODENUM

Node number associated with the specified element *ELNUM*.

PHI

Signed normal distance of the node from the crack.

PSI

Signed normal distance of the node from the crack tip (or crack front). Used only in the singularity-based XFEM method.

Notes

Issue the **XFDATA (p. 2192)** command multiple times as needed to specify nodal level set values for all nodes of an element.

This command is valid in PREP7 (**/PREP7 (p. 1479)**) only.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

XFENRICH, *EnrichmentID*, *CompName*, *MAT_ID*, *Method*, *RADIUS*, *SNAPTOLER*

Defines parameters associated with crack propagation using XFEM

PREP7 (p. 22): Special Purpose (p. 37)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

EnrichmentID

An alphanumeric name assigned to identify the enrichment. The name can contain up to 32 characters and must begin with an alphabetic character. Alphabetic characters, numbers, and underscores are valid.

CompName

Name of the element set component for which initial cracks are defined and possibly propagated.

MAT_ID

Material ID number referring to cohesive zone material behavior on the initial crack. If 0 or not specified, the initial crack is assumed to be free of cohesive zone behavior. Used only with the phantom-node XFEM method (*Method*).

Method

PHAN -- Use phantom-node-based XFEM (default).

SING -- Use singularity-based XFEM.

RADIUS

Radius defining the region around the crack tip encompassing the set of elements to be influenced by the crack-tip singularity effects. Default = 0.0. Used only in singularity-based XFEM.

SNAPTOLER

Snap tolerance to snap the crack tip to the closest crack face along the extension direction. Default = 1.0E-6. Used only in singularity-based XFEM.

Notes

If *MAT_ID* is specified, the cohesive zone behavior is described by the bilinear cohesive law.

If issuing multiple **XFENRICH (p. 2193)** commands, the element components (*CompName*) should not intersect (that is, the element components should not have any common elements between them).

When multiple **XFENRICH (p. 2193)** commands are issued in an analysis, combining the phantom-node-based method (*Method* = PHAN) and the singularity-based method (*Method* = SING) is not valid. Only one XFEM method per analysis is allowed.

This command is valid in PREP7 (**/PREP7 (p. 1479)**) only.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

This command cannot be accessed from a menu.

XFLIST, *EnrichmentID*

Lists enrichment details and associated crack information

PREP7 (p. 22): Special Purpose (p. 37)
SOLUTION (p. 38): Misc Loads (p. 45)

Valid Products: (p. 2) Pro | Premium | **Enterprise** | **PrepPost** | **Solver** | AS add-on

***EnrichmentID* or (blank)**

Name of the enrichment specified via the associated **XFENRICH (p. 2193)** command. Specifying *EnrichmentID* is optional.

If no value (or ALL) is specified, the command lists all enrichments and associated data.

Notes

This command is valid in PREP7 (**/PREP7 (p. 1479)**) and SOLUTION (**/SOLU (p. 1821)**).

Menu Paths

This command cannot be accessed from a menu.

/XFRM, *LAB*, *X1*, *Y1*, *Z1*, *X2*, *Y2*, *Z2*

Controls the centroid or the axis of dynamic rotation.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

LAB

The location or entity (centroid) used to define the center or axis of rotation.

NODE

If NODE is chosen for the center of rotation, the node number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two nodes between which a line is drawn to determine the axis. The remaining arguments are ignored.

ELEMENT

If ELEMENT is chosen for the center of rotation, the element number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two elements between which a line is drawn to determine the axis. The remaining arguments are ignored.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

KP

If KP is chosen for the center of rotation, the keypoint number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two keypoints between which a line is drawn to determine the axis. The remaining arguments are ignored.

LINE

If LINE is chosen for the center of rotation, the line number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two lines between which a line is drawn to determine the axis. The remaining arguments are ignored.

AREA

If AREA is chosen for the center of rotation, the area number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two areas between which a line is drawn to determine the axis. The remaining arguments are ignored.

VOLUME

If VOLUME is chosen for the center of rotation, the volume number will be *X1*. If the rotation is to be about an axis, then *X1* and *Y1* define the two volumes between which a line is drawn to determine the axis. The remaining arguments are ignored.

XYZ

If XYZ is chosen for the center of rotation, the location of that center is determined by the coordinates *X1*, *Y1*, *Z1*. If values are specified for *X2*, *Y2*, *Z2*, then the axis of rotation will be about the line between those two points.

OFF

If *LAB* = OFF, DEFAULT, FOCUS or if no value is specified, then the center of rotation is set at the FOCUS point, as defined by the **/FOCUS (p. 771)** command.

X1

The entity number or X coordinate for the center of rotation.

Y1

The entity number or Y coordinate for the center of rotation.

Z1

The Z coordinate for the center of rotation.

X2

The X coordinate for the axis of rotation.

Y2

The Y coordinate for the axis of rotation.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z2

The Z coordinate for the axis of rotation.

Command Default

Issuing **/XFRM (p. 2194)**, with no *LAB* defined sets the center of rotation at the focal point specified by the **/FOCUS (p. 771)** command.

Notes

The **/XFRM (p. 2194)** command is active only when the cumulative rotation key is specified ON for the **/ANGLE (p. 140)** command (*KINCR* = 1). This command affects dynamic manipulations only.

For center rotation, the middle mouse button will rotate the model about the screen Z axis and the right mouse button will rotate the model about the screen X and Y axis.

For rotation about an axis, the middle mouse button will rotate the model about the defined axis of rotation and the right mouse button will be deactivated.

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>ViewSettings>Rotational Center>ByPick

***XPL**,*Action*,*Val1*,*Val2*

Accesses content of a Mechanical APDL file.

APDL (p. 19): Matrix Operations (p. 21)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

Action

Specifies the action for scanning file content (no default). Valid labels are:

OPEN

Open the specified file.

CLOSE

Close the specified file.

LIST

List the records at the current level in the hierarchy of records.

WHERE

Display the current location in the tree.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

STEP

Step down in the tree of records.

UP

Go up in the tree of records.

READ

Read a record into an APDL Math object.

INFO

Display information from a record.

GOTO

Move directly to a given place in the tree of records (this avoids multiple calls to the STEP and UP options).

MARK

Mark a set of records of the current file; the asterisk (*) character can be used to specify multiple branches/records.

COPY

Copy the current file to a new one, ignoring the marked records.

SAVE

Save the current file, ignoring the marked records.

Val1*, *Val2

Additional input. The meanings of *Val1* and *Val2* vary depending on the specified *Action*, as described in the table below.

Action **Label**	***Val1***	***Val2***
OPEN	Name of file[a] to open (no default)	(not used)
CLOSE	Name of file[a] to close (default is last file opened by *Action* = OPEN)	(not used)
LIST	Number of levels in the tree of records (default = 1)	(not used)
UP	Number of levels in the tree of records (default = 1), or TOP to go directly to the top of the tree	(not used)
STEP	Name of the record of interest (no default)	(not used)
INFO	Name of the record of interest (no default), or * for all records	(not used)

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Action **Label**	***Val1***	***Val2***
READ	Name of the record of interest (no default)	Name of the APDL Math object to create (no default)
GOTO	The complete PATH to the new location Example: `*XPL,GOTO,RST::DSI::SET1`	(not used)
MARK	The complete path to the record(s); use (*) character to select multiple records (see example in next section)	(not used)
COPY	Name of the new file[a]	If blank (default), marked records are not copied to the new file. Valid inputs are: COMP - records are compressed using sparsification UNCOMP - records are uncompressed

[a] File name is case-sensitive with a 32-character maximum.

Notes

The ***XPL** command enables you to explore the contents of a Mechanical APDL file. Use this command to traverse up and down the tree structure of the specified file and review what is in the file. Files that can be scanned include `.rst`, `.mode`, `.full`, `.cms`, and `.sub` files. For more information and examples, see Using APDL to List File Structure and Content in the *Ansys Parametric Design Language Guide*.

The command ***XPL**,READ generates either a ***VEC (p. 2085)** or a ***DMAT (p. 551)** object according to the record type. You do not have to create the APDL Math object before issuing this command.

This command is valid in any processor.

Menu Paths

This command cannot be accessed from a menu.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/XRANGE, *XMIN*, *XMAX*

Specifies a linear abscissa (X) scale range.

GRAPHICS (p. 16): Graphs (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

XMIN

Minimum abscissa scale value.

XMAX

Maximum abscissa scale value.

Command Default

Automatically select X-range scale to include all data being displayed.

Notes

Specifies a linear abscissa (X) scale range for the line graph display. Use **/XRANGE (p. 2199)**,DEFAULT to return to automatic scaling.

Automatic scaling will often yield inappropriate range values for logarithmic scales (**/GROPT (p. 870)**, LOGX).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

XVAR, *N*

Specifies the X variable to be displayed.

POST26 (p. 58): Display (p. 60)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

N

X variable number:

0 or 1

Display **PLVAR (p. 1444)** values vs. time (or frequency).

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

n

Display **PLVAR (p. 1444)** values vs. variable *n* (2 to *NV* (**NUMVAR (p. 1302)**)).

1

Interchange time and **PLVAR (p. 1444)** variable numbers with time as the curve parameter. **PLVAR (p. 1444)** variable numbers are displayed uniformly spaced along X-axis from position 1 to 10.

Command Default

Use time or frequency.

Notes

Defines the X variable (displayed along the abscissa) against which the Y variable(s) (**PLVAR (p. 1444)**) are to be displayed.

Menu Paths

Main Menu>TimeHist Postpro>Settings>Graph

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y Commands

/YRANGE, *YMIN*, *YMAX*, *NUM*

Specifies a linear ordinate (Y) scale range.

GRAPHICS (p. 16): Graphs (p. 18)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

YMIN

Minimum ordinate scale value.

YMAX

Maximum ordinate scale value.

NUM

Y-axis number to which range applies (defaults to 1). Valid numbers are 1 to 3 for **/GRTYP (p. 874)**,2 and 1 to 6 for **/GRTYP (p. 874)**,3. If ALL, apply to all Y-axes.

Command Default

Automatically select Y-range scale to include all data being displayed.

Notes

Specifies a linear ordinate (Y) scale range for the line graph display. Use **/YRANGE (p. 2201)**,DEFAULT to return to automatic scaling. For multiple Y-axes graphs (**/GRTYP (p. 874)**), see **/GROPT (p. 870)**, ASCAL to automatically scale the additional Y-axes.

Automatic scaling will often yield inappropriate range values for logarithmic scales (**/GROPT (p. 870)**, LOGY).

This command is valid in any processor.

Menu Paths

Utility Menu>PlotCtrls>Style>Graphs>Modify Axes

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Z Commands

/ZOOM, *WN*, *Lab*, *X1*, *Y1*, *X2*, *Y2*

Zooms a region of a display window.

GRAPHICS (p. 16): Views (p. 17)

Valid Products: (p. 2) **Pro** | **Premium** | **Enterprise** | **PrepPost** | **Solver** | AS add-on

WN

Window number to which command applies (defaults to 1).

Lab

Label to define the desired type of zoom:

OFF

Turns zoom off (refits image of entire model to the window).

BACK

Goes back to previous zoom setting (five successive back ups, maximum).

SCRN

Interprets X1,Y1 as the screen coordinates of the center of a square zoom region; X2,Y2 as the screen coordinates of a point on one side of that square.

RECT

Interprets X1,Y1 and X2,Y2 as the screen coordinates of two opposite corners of a rectangular zoom region.

Notes

Zooms (centers and magnifies) the specified region of a display window. **/ZOOM (p. 2203)** will operate on a display that has been formed by an explicit graphics action command (**APLOT (p. 169)**, **EPLOT (p. 648)**, etc.). **/ZOOM (p. 2203)** has no effect on an "immediate" graphics display. When **/ZOOM (p. 2203)** is executed, the display is automatically replotted such that the specified zoom region is centered and sized to fill the window.

Auto resizing is disabled when you issue the **/ZOOM (p. 2203)** command. To restore auto resizing, issue the **/AUTO (p. 207)** command, or select **FIT** from the Pan, Zoom, Rotate box.

This command is valid in any processor.

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Menu Paths

Utility Menu>PlotCtrls>Pan, Zoom, Rotate

Release 2024 R1 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.