



Material Reference



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 2020 R1
January 2020

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

Copyright and Trademark Information

© 2020 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. Introduction to Material Models	1
1.1. Material Models for Displacement Applications	1
1.2. Material Models for Temperature Applications	2
1.3. Material Models for Electromagnetic Applications	2
1.4. Material Models for Coupled Applications	3
1.5. Material Parameters	3
1.6. How Material Properties Are Evaluated	3
2. Material Model Element Support	5
3. Linear Material Properties	13
3.1. Defining Linear Material Properties	13
3.1.1. Defining Linear Material Properties Using Tabular Input	16
3.2. Stress-Strain Relationships	18
3.3. Anisotropic Elasticity	19
3.4. Damping	20
3.5. Thermal Expansion	21
3.6. Emissivity	22
3.7. Specific Heat	22
3.8. Film Coefficients	22
3.9. Temperature Dependency	23
4. Nonlinear Material Properties	25
4.1. Understanding Material Data Tables	26
4.2. Experimental Data	26
4.3. Porous Elasticity	26
4.3.1. Defining the Porous Elasticity Model	28
4.4. Rate-Independent Plasticity	28
4.4.1. Understanding the Plasticity Models	29
4.4.1.1. Nomenclature	30
4.4.1.2. Strain Decomposition	31
4.4.1.3. Yield Criterion	31
4.4.1.4. Flow Rule	32
4.4.1.5. Hardening	33
4.4.1.6. Large Deformation	34
4.4.1.7. Output	34
4.4.1.8. Resources	35
4.4.2. Isotropic Hardening	36
4.4.2.1. Yield Criteria and Plastic Potentials	37
4.4.2.1.1. Von Mises Yield Criterion	37
4.4.2.1.2. Hill Yield Criterion	38
4.4.2.1.2.1. Separated Hill Potentials for Plasticity and Creep	40
4.4.2.2. General Isotropic Hardening Classes	40
4.4.2.2.1. Bilinear Isotropic Hardening	40
4.4.2.2.1.1. Defining the Bilinear Isotropic Hardening Model	41
4.4.2.2.2. Multilinear Isotropic Hardening	42
4.4.2.2.2.1. Defining the Multilinear Isotropic Hardening Model	42
4.4.2.2.3. Nonlinear Isotropic Hardening	43
4.4.2.2.3.1. Power Law Nonlinear Isotropic Hardening	43
4.4.2.2.3.2. Voce Law Nonlinear Isotropic Hardening	44
4.4.2.2.4. Isotropic Hardening Static Recovery	45
4.4.2.2.4.1. Defining the Isotropic Static Recovery	46
4.4.3. Kinematic Hardening	46

4.4.3.1. Yield Criteria and Plastic Potentials	47
4.4.3.2. General Kinematic Hardening Classes	47
4.4.3.2.1. Bilinear Kinematic Hardening	47
4.4.3.2.1.1. Defining the Bilinear Kinematic Hardening Model	48
4.4.3.2.2. Multilinear Kinematic Hardening	49
4.4.3.2.2.1. Defining the Multilinear Kinematic Hardening Model	51
4.4.3.2.2.2. Specifying the Constants	51
4.4.3.2.3. Nonlinear Kinematic Hardening	51
4.4.3.2.3.1. Defining the Nonlinear Kinematic Hardening Model	52
4.4.3.3. Kinematic Hardening Static Recovery	53
4.4.3.3.1. Defining the Kinematic Static Recovery	53
4.4.4. Drucker-Prager	54
4.4.4.1. Classic Drucker-Prager	54
4.4.4.1.1. Defining the Classic Drucker-Prager Model	55
4.4.4.2. Extended Drucker-Prager (EDP)	55
4.4.4.2.1. EDP Yield Criteria Forms	55
4.4.4.2.1.1. Linear Form	55
4.4.4.2.1.2. Power Law Form	56
4.4.4.2.1.3. Hyperbolic Form	57
4.4.4.2.2. EDP Plastic Flow Potentials	58
4.4.4.2.2.1. Linear Form	58
4.4.4.2.2.2. Power Law Form	58
4.4.4.2.2.3. Hyperbolic Form	59
4.4.4.2.3. Plastic Strain Increments for Flow Potentials	59
4.4.4.2.4. Example EDP Material Model Definitions	59
4.4.4.3. Extended Drucker-Prager Cap	60
4.4.4.3.1. Defining the EDP Cap Yield Criterion and Hardening	63
4.4.4.3.2. Defining the EDP Cap Plastic Potential	64
4.4.4.3.3. Example EDP Cap Material Model Definition	64
4.4.5. Gurson	64
4.4.5.1. Void Volume Fraction	65
4.4.5.2. Hardening	67
4.4.5.3. Defining the Gurson Material Model	67
4.4.5.3.1. Defining the Gurson Base Model	68
4.4.5.3.2. Defining Stress- or Strain-Controlled Nucleation	68
4.4.5.3.3. Defining the Void Coalescence Behavior	68
4.4.5.3.4. Example Gurson Model Definition	69
4.4.6. Cast Iron	69
4.4.6.1. Defining the Cast Iron Material Model	74
4.4.7. Crushable Foam	75
4.4.7.1. Defining the Crushable Foam Material Model	77
4.4.7.2. Resources	78
4.5. Rate-Dependent Plasticity (Viscoplasticity)	78
4.5.1. Perzyna and Peirce Options	78
4.5.2. Exponential Visco-Hardening (EVH) Option	79
4.5.3. Anand Option	80
4.5.4. Creep	81
4.5.4.1. Implicit Creep Equations	82
4.6. Hyperelasticity	83
4.6.1. Arruda-Boyce Hyperelasticity	84
4.6.2. Blatz-Ko Foam Hyperelasticity	85
4.6.3. Extended Tube Hyperelasticity	85

4.6.4. Gent Hyperelasticity	86
4.6.5. Mooney-Rivlin Hyperelasticity	86
4.6.6. Neo-Hookean Hyperelasticity	88
4.6.7. Ogden Hyperelasticity	88
4.6.8. Ogden Compressible Foam Hyperelasticity	89
4.6.9. Polynomial Form Hyperelasticity	90
4.6.10. Response Function Hyperelasticity	91
4.6.11. Yeoh Hyperelasticity	92
4.6.12. Special Hyperelasticity	93
4.6.12.1. Anisotropic Hyperelasticity	93
4.6.12.2. Bergstrom-Boyce Material	94
4.6.12.3. Mullins Effect	95
4.6.12.4. User-Defined Hyperelastic Material	96
4.7. Viscoelasticity	96
4.7.1. Viscoelastic Formulation	97
4.7.1.1. Small Deformation	97
4.7.1.2. Small Strain with Large Deformation	99
4.7.1.3. Large Deformation	100
4.7.1.4. Stored Energy and Dissipation	100
4.7.2. Time-Temperature Superposition	102
4.7.2.1. Williams-Landel-Ferry Shift Function	102
4.7.2.2. Tool-Narayanawamy Shift Function	102
4.7.2.3. User-Defined Shift Function	104
4.7.3. Harmonic Viscoelasticity	104
4.7.3.1. Prony Series Complex Modulus	105
4.7.3.2. Experimental Data Complex Modulus	105
4.7.3.3. Frequency-Temperature Superposition	106
4.7.3.4. Stress	107
4.8. Microplane	107
4.8.1. Microplane Modeling	107
4.8.1.1. Discretization	109
4.8.1.2. Implicit Gradient Regularization	110
4.8.1.2.1. Identifying the Nonlocal Interaction-Range Parameter c	110
4.8.2. Microplane Material Models	111
4.8.2.1. Elastic Damage Microplane Material Models	111
4.8.2.1.1. Nonregularized Elastic Damage Microplane Material Model	112
4.8.2.1.2. Regularized Elastic Damage Microplane Material Model	113
4.8.2.1.3. Defining the Elastic Damage Microplane Material Models	114
4.8.2.1.4. Output for the Elastic Damage Microplane Material Models	116
4.8.2.2. Coupled Damage-Plasticity Microplane Model	116
4.8.2.2.1. Smooth Three-Surface Microplane Drucker-Prager Cap Yield Function	117
4.8.2.2.2. Damage Evolution	118
4.8.2.2.3. Implicit Gradient Regularization	120
4.8.2.2.4. Coupled Damage-Plasticity Microplane Model Parameters	121
4.8.2.2.5. Defining the Coupled Damage-Plasticity Microplane Model	122
4.8.2.2.6. Identifying Coupled Damage-Plasticity Microplane Model Parameters	124
4.8.2.2.7. Coupled Damage-Plasticity Microplane Damage Output	125
4.8.2.3. Solution Control for the Regularized Microplane Damage Models	125
4.8.3. Learning More About Microplane Material Modeling	126
4.9. Geomechanics	126
4.9.1. Understanding the Material Models for Geomechanics	127
4.9.1.1. Guidance for Using the Geomechanics Material Models	128

4.9.1.2. Material Solution Option	129
4.9.1.3. Nomenclature	129
4.9.1.4. Resources	130
4.9.2. Cam-clay	131
4.9.2.1. Yield Surface	131
4.9.2.2. Defining the Cam-clay Model	135
4.9.3. Mohr-Coulomb	136
4.9.3.1. Understanding the Mohr-Coulomb Model	136
4.9.3.1.1. Yield Surface	136
4.9.3.1.2. Plastic Flow Potentials	141
4.9.3.2. Defining the Mohr-Coulomb Model	142
4.9.4. Jointed Rock	145
4.9.4.1. Mohr-Coulomb Base Material	146
4.9.4.2. Anisotropic Mohr-Coulomb Joints	146
4.9.4.3. Defining the Jointed Rock Model	150
4.9.4.3.1. Part 1: Defining the Base Material	150
4.9.4.3.2. Part 2: Defining the Joints	151
4.9.5. Drucker-Prager Concrete	153
4.9.5.1. Yield Surfaces and Flow Potentials	153
4.9.5.1.1. Tension and Tension-Compression Surfaces	153
4.9.5.1.2. Compression Surface	154
4.9.5.2. Defining the Drucker-Prager Concrete Model	156
4.9.5.2.1. Hardening, Softening and Dilatation (HSD) Behavior	157
4.9.5.2.1.1. Exponential HSD Model (TB,CONCR,,,,HSD2)	158
4.9.5.2.1.2. Steel Reinforcement HSD Model (TB,CONCR,,,,HSD4)	160
4.9.5.2.1.3. Fracture Energy HSD Model (TB,CONCR,,,,HSD5)	162
4.9.5.2.1.4. Linear HSD Model (TB,CONCR,,,,HSD6)	163
4.9.5.2.2. Representing Weak Joints in Concrete	165
4.9.5.2.3. Example Drucker-Prager Concrete Definitions	166
4.9.6. Menetrey-Willam	167
4.9.6.1. Yield Surfaces and Flow Potentials	167
4.9.6.2. Defining the Menetrey-Willam Model	170
4.9.6.3. Hardening and Softening Functions	170
4.9.6.3.1. Linear Softening	171
4.9.6.3.2. Exponential Softening	172
4.9.6.4. Example Menetrey-Willam Definitions	173
4.9.6.5. References	174
4.10. Porous Media	174
4.10.1. Fluid Flow and Permeability	175
4.10.1.1. Permeability	175
4.10.2. Porous Media Mechanics	175
4.10.2.1. Momentum-Balance Equation	176
4.10.2.2. Mass-Balance (Continuity) Equation for the Pore-Fluid Phase	177
4.10.2.2.1. Compressibility Parameter	177
4.10.2.2.2. Biot Coefficient	178
4.10.2.2.3. Effective Stress	178
4.10.2.2.4. Free-Strain Rate	178
4.10.2.3. Heat-Transfer Equation	179
4.10.3. Porous Media Material Properties	180
4.10.3.1. Permeability (TB,PM,,,,TBOPT = PERM)	180
4.10.3.2. Biot Coefficient (TB,PM,,,, TBOPT = BIOT)	181
4.10.3.3. Solid Property (TB,PM,,,, TBOPT = SP)	181

4.10.3.4. Fluid Property (TB,PM,,,, TBOPT = FP)	182
4.10.3.5. Degree of Saturation (TB,PM,,,, TBOPT = DSAT)	182
4.10.3.6. Relative Permeability (TB,PM,,,, TBOPT = RPER)	182
4.10.3.7. Gravity Magnitude (TB,PM,,,, TBOPT = GRAV)	182
4.10.3.8. Units of Permeability	183
4.10.4. Heat-Transfer Properties	183
4.10.5. Thermal-Expansion Material Properties	183
4.10.6. Transient vs. Static Analysis	183
4.10.7. Partially Saturated Porous Media Flow and Coupled-Pore-Pressure-Thermal (CPT) Damp- ing	183
4.10.8. Additional Resources	184
4.11. Gasket	184
4.12. Swelling	186
4.13. Shape Memory Alloy (SMA)	188
4.13.1. SMA Model for Superelasticity	189
4.13.1.1. Constitutive Model for Superelasticity	189
4.13.1.2. Material Parameters for the Superelastic SMA Material Model	192
4.13.2. SMA Material Model with Shape Memory Effect	193
4.13.2.1. The Constitutive Model for Shape Memory Effect	193
4.13.2.2. Material Parameters for the Shape Memory Effect Option	195
4.13.3. Result Output of Solution Variables	196
4.13.4. Element Support for SMA	196
4.13.5. Learning More About Shape Memory Alloy	197
4.14. Multilinear Elasticity	197
4.15. MPC184 Joint	199
4.15.1. Linear Elastic Stiffness and Damping Behavior	199
4.15.2. Nonlinear Elastic Stiffness and Damping Behavior	200
4.15.2.1. Specifying a Function Describing Nonlinear Stiffness Behavior	202
4.15.3. Frictional Behavior	202
4.16. Contact Friction	204
4.16.1. Isotropic Friction	205
4.16.2. Orthotropic Friction	205
4.16.3. Redefining Friction Between Load Steps	206
4.16.4. User-Defined Friction	207
4.17. Contact Interaction	207
4.17.1. Interaction Options for General Contact Definitions	207
4.17.2. User-Defined Interaction	209
4.18. Cohesive Material Law	209
4.18.1. Exponential Cohesive Zone Material for Interface Elements and Contact Elements	210
4.18.2. Bilinear Cohesive Zone Material for Interface Elements and Contact Elements	210
4.18.3. Viscous Regularization of Cohesive Zone Material for Interface Elements and Contact Ele- ments	211
4.18.4. Cohesive Zone Material for Contact Elements	211
4.18.5. Post-Debonding Behavior at the Contact Interface	213
4.19. Contact Surface Wear	214
4.19.1. Archard Wear Model	214
4.19.2. User-Defined Wear Model	215
4.20. Custom Material Models	215
4.20.1. Creating a User-Defined Material Model	215
4.20.1.1. User-Defined Material Model (UserMat)	215
4.20.1.2. User-Defined Thermal Material Model (UserMatTh)	216
4.20.2. User-Defined Cohesive Material (UserCZM)	216

4.20.3. Using State Variables with User-Defined Materials	217
4.20.3.1. Using State Variables with the UserMat Subroutine	217
4.20.3.2. Using State Variables with the UserMatTh Subroutine	217
4.20.3.3. Using State Variables with the UserCZM Subroutine	218
4.21. Material Strength Limits	218
4.22. Material Damage	220
4.22.1. Damage Initiation Criteria	220
4.22.2. Damage Evolution Law	221
4.22.2.1. Predicting Post-Damage Degradation of Brittle Anisotropic Materials	223
4.22.2.1.1. Damage Modes	226
4.23. Material Damping	227
4.23.1. Material-Dependent Alpha and Beta Damping (Rayleigh Damping)	227
4.23.1.1. Defining Material-Dependent Alpha and Beta Damping (Rayleigh Damping)	228
4.23.1.1.1. Defining Rayleigh Damping via MP Commands	228
4.23.1.1.2. Defining Rayleigh Damping via TB Commands	229
4.23.1.1.3. Comparing Feature Support: MP - vs. TB -Defined Damping	230
4.23.2. Material-Dependent Structural Damping	230
4.23.2.1. Using MP to Define Material-Dependent Structural Damping Coefficients	231
4.23.2.2. Using TB to Define Material-Dependent Structural Damping Coefficients	231
4.23.2.3. Comparing Structural Damping Coefficient Options	232
4.23.3. Viscoelastic Material Damping (Harmonic Viscoelasticity)	232
5. Multiphysics Material Properties	233
5.1. Acoustics	233
5.1.1. Perforated Media	233
5.1.1.1. Equivalent Fluid Model of Perforated Media	233
5.1.1.1.1. Johnson-Champoux-Allard Equivalent Fluid Model of Perforated Media	234
5.1.1.1.2. Delany-Bazley Equivalent Fluid Model of Perforated Media	235
5.1.1.1.3. Miki Equivalent Fluid Model of Perforated Media	235
5.1.1.1.4. Complex Impedance and Propagating-Constant Equivalent Fluid Model of Perforated Media	236
5.1.1.1.5. Complex Density and Velocity Equivalent Fluid Model of Perforated Media	237
5.1.1.2. Poroelastic Acoustic Material	237
5.1.1.3. Transfer Admittance Matrix Models	238
5.1.1.3.1. Transfer Admittance Matrix Model of Perforated Media	238
5.1.1.3.2. Transfer Admittance Matrix Model of a Square or Hexagonal Grid Structure	239
5.1.2. Acoustic Frequency-Dependent Materials	240
5.1.3. Low Reduced Frequency (LRF) Model of Acoustic Viscous-Thermal Media	240
5.1.3.1. Thin Layer	240
5.1.3.2. Tube with Rectangular Cross-Section	241
5.1.3.3. Tube with Circular Cross-Section	241
5.1.4. Diffusion Properties for Room Acoustics	241
5.2. Fluids	242
5.3. Electricity and Magnetism	243
5.3.1. Piezoelectricity	243
5.3.2. Piezoresistivity	244
5.3.3. Magnetism	245
5.3.4. Anisotropic Electric Permittivity	246
5.3.5. Anisotropic Viscosity	247
5.3.6. Anisotropic Elastic Loss Tangent	249
5.3.7. Anisotropic Dielectric Loss Tangent	250
5.4. Migration Model	251
5.4.1. Diffusion Flux and Chemical Potential	251

5.4.2. Atomic Flux Option (<i>TBOPT</i> = 0)	252
5.4.3. Vacancy Flux Option (<i>TBOPT</i> = 1)	254
5.5. Thermal Properties	257
5.5.1. Thermal Conductivity (<i>TBOPT</i> = COND)	257
5.5.2. Specific Heat (<i>TBOPT</i> = SPHT)	258
5.5.3. Fluid Specific Heat (<i>TBOPT</i> = FLSPHT)	258
6. Material Curve-Fitting	259
6.1. Hyperelastic Material Curve-Fitting	259
6.1.1. Understanding the Hyperelastic Material Curve-Fitting Process	260
6.1.2. Step 1. Prepare Hyperelastic Experimental Data	260
6.1.3. Step 2. Input the Hyperelastic Experimental Data	261
6.1.3.1. Syntax and Arguments	262
6.1.4. Step 3. Select a Hyperelastic Material Model Option	262
6.1.4.1. Syntax and Arguments	263
6.1.5. Step 4. Initialize the Hyperelastic Coefficients	263
6.1.5.1. Syntax and Arguments	264
6.1.6. Step 5. Specify Hyperelastic Control Parameters and Solve	265
6.1.6.1. Syntax and Arguments	265
6.1.7. Step 6. Plot Your Hyperelastic Experimental Data and Analyze	266
6.1.7.1. Plotting	266
6.1.7.2. Review/Verify	266
6.1.8. Step 7. Write Hyperelastic Curve-Fitting Data to the Database	266
6.1.8.1. Syntax and Arguments	266
6.2. Viscoelastic Material Curve-Fitting	267
6.2.1. Understanding the Viscoelastic Material Curve-Fitting Process	267
6.2.2. Step 1. Prepare Viscoelastic Experimental Data	268
6.2.3. Step 2. Input the Viscoelastic Data	269
6.2.3.1. Syntax and Arguments	269
6.2.4. Step 3. Select a Viscoelastic Material Model Option	270
6.2.4.1. Syntax and Arguments	270
6.2.5. Step 4. Initialize the Viscoelastic Coefficients	270
6.2.5.1. Syntax and Arguments	272
6.2.6. Step 5. Specify Viscoelastic Control Parameters and Solve	273
6.2.6.1. Viscoelastic Parameter Scaling	273
6.2.6.2. Temperature-Dependent Solutions Using the Shift Function	274
6.2.6.3. Temperature-Dependent Solutions Without the Shift Function	274
6.2.6.4. Syntax and Arguments	275
6.2.7. Step 6. Plot the Viscoelastic Experimental Data and Analyze	276
6.2.7.1. Analyze Your Curves for Proper Fit	276
6.2.8. Step 7. Write Viscoelastic Curve-Fitting Data to the Database	276
6.2.8.1. Syntax and Arguments	277
6.3. Chaboche Material Curve-Fitting	277
6.3.1. Understanding the Chaboche Material Curve-Fitting Process	278
6.3.2. Step 1. Prepare Chaboche Experimental Data	279
6.3.3. Step 2. Input the Chaboche Experimental Data	280
6.3.3.1. Syntax and Arguments	280
6.3.4. Step 3. Select a Chaboche Material Model Option	280
6.3.4.1. Syntax and Arguments	280
6.3.5. Step 4. Initialize the Chaboche Coefficients	281
6.3.5.1. Including Isotropic Hardening Models with Chaboche Kinematic Hardening	281
6.3.5.2. General Process for Initializing MISO Option Coefficients	282
6.3.5.2.1. Syntax and Arguments for Coefficient Initialization	282

6.3.6. Step 5. Specify Chaboche Control Parameters and Solve	283
6.3.6.1. Chaboche Parameter Scaling	284
6.3.6.2. Temperature-Dependent Solutions	284
6.3.6.3. Syntax and Arguments	284
6.3.7. Step 6. Plot the Chaboche Experimental Data and Analyze	285
6.3.7.1. Analyzing Your Curves for Proper Fit	285
6.3.8. Step 7. Write Chaboche Curve-Fitting Data to the Database	286
6.4. Creep Material Curve-Fitting	286
6.4.1. Understanding the Creep Material Curve-Fitting Process	286
6.4.2. Step 1. Prepare Creep Experimental Data	287
6.4.3. Step 2. Input the Creep Experimental Data	288
6.4.3.1. Syntax and Arguments	288
6.4.4. Step 3. Select a Creep Material Model Option	289
6.4.4.1. Syntax and Arguments	289
6.4.5. Step 4. Initialize the Creep Coefficients	290
6.4.5.1. Syntax and Arguments	290
6.4.6. Step 5. Specify Creep Control Parameters and Solve	291
6.4.6.1. Syntax and Arguments	291
6.4.6.2. Creep Parameter Scaling	292
6.4.7. Step 6. Plot the Creep Experimental Data and Analyze	293
6.4.7.1. Analyze Your Curves for Proper Fit	293
6.4.8. Step 7. Write Creep Curve-Fitting Data to the Database	293
6.4.8.1. Syntax and Arguments	293
6.4.9. Hints for Curve-Fitting Creep Models	294
7. Material Model Combinations	297
8. Understanding Field Variables	301
8.1. Predefined Field Variables	301
8.1.1. Defining Friction	304
8.1.2. Defining Young's Modulus as a Function of Global X,Y	304
8.2. User-Defined Field Variables	304
8.2.1. Subroutine for Editing Field Variables	305
8.3. Data Processing	306
8.4. Logarithmic Interpolation and Scaling	307
8.5. Interpolation Algorithms	307
8.5.1. Simple Linear Interpolation	308
8.5.1.1. Simple Linear Algorithm (One-Dimensional)	308
8.5.1.2. Simple Linear Algorithm (Two-Dimensional)	309
8.5.2. Multidimensional Interpolation	310
8.5.2.1. Linear-Multivariate Algorithm	311
8.5.3. Evaluating Interpolation Algorithm Results	313
8.5.4. Material Model Support for Interpolation	314
8.5.5. Reference	314
9. GUI-Inaccessible Material Properties	315

List of Figures

4.1. Stress-Strain Curve for an Elastic-Plastic Material	29
4.2. Yield Surface in Principal Stress Space	32
4.3. Plastic Strain Flow Rule	32
4.4. Isotropic Hardening of the Yield Surface	33
4.5. Kinematic Hardening of the Yield Surface	34
4.6. Yield Surface for von Mises Yield Criterion	38
4.7. Stress vs. Total Strain (a), and Stress vs. Plastic Strain (b) for Bilinear Isotropic Hardening	41
4.8. Stress vs. Total Strain for Multilinear Isotropic Hardening	42
4.9. Stress vs. Plastic Strain for Voce Hardening	44
4.10. Stress vs. Total Strain (a), and Stress vs. Plastic Strain (b) for Bilinear Kinematic Hardening	48
4.11. Stress vs. Total Strain for Multilinear Kinematic Hardening	50
4.12. Power Law Criterion in the Meridian Plane	56
4.13. Hyperbolic and Linear Criterion in the Meridian Plane	57
4.14. Yield Surface for the Cap Criterion	61
4.15. Relationship Between ε_v^p and X_0	62
4.16. Growth, Nucleation, and Coalescence of Voids at Microscopic Scale	65
4.17. Cast Iron Yield Surface	71
4.18. Composite Cast Iron Yield Surface in 2-D Principal Stress	71
4.19. Comparison of Cast Iron Yield Surfaces	72
4.20. Cast Iron Yield Surface with Tension Rounding	72
4.21. Crushable Foam Yield Surface	75
4.22. Generalized Maxwell Solid in One Dimension	97
4.23. Sphere Discretization by 42 Microplanes	109
4.24. Damage Parameter d^{mic} Depending on the Equivalent Strain	112
4.25. Stress-strain Behavior at Uniaxial Tension	113
4.26. Smooth Three-Surface Microplane Cap Yield Function	117
4.27. Effect of Damage Split on Tension-Compression Cyclic Loading Response	119
4.28. Modified Cam-clay Yield Surface Plotted in Principal Stresses	131
4.29. Modified Cam-clay Yield Surface in the Octahedral Plane	132
4.30. Modified Cam-clay Yield Surface in the q-p Plane	132
4.31. Linear Relationship Between Void Ratio and Natural Log of Pressure	134
4.32. Hardening and Softening Behavior	134
4.33. Mohr-Coulomb Yield Surface as Shear vs. Mean Stress	137
4.34. Mohr-Coulomb Yield Surface in Principal Stresses	138
4.35. Mohr-Coulomb with Tension-Failure Yield Surface in Principal Stress	139
4.36. Flow Potentials in the Octahedral Plane	142
4.37. Joint Failure Surface Superimposed on Mohr-Coulomb with Tension-Cutoff Yield Surface	147
4.38. Composite Yield Surface of Jointed Rock with Mohr-Coulomb Base Material	148
4.39. Reorienting the Failure Plane	149
4.40. Composite Surface with Drucker-Prager Compression and Drucker-Prager Tension Yield Surfaces	155
4.41. 2-D Yield Surfaces Showing Drucker-Prager and Rankine Surfaces	156
4.42. Exponential Softening in Compression and Tension	159
4.43. Steel Reinforcement Softening in Compression and Tension	160
4.44. Steel Reinforcement Dilatancy in Tension	161
4.45. Fracture Energy Softening in Compression and Tension	163
4.46. Linear Softening in Compression and Tension	164
4.47. Yield Surface for the Menetrey-Willam Model	169
4.48. Linear Softening in Compression and Tension	171
4.49. Exponential Softening in Compression and Tension	173
4.50. Pseudoelasticity (PE) and Shape Memory Effect (SME)	188

4.51. Typical Superelasticity Behavior	190
4.52. Idealized Stress-Strain Diagram of Superelastic Behavior	191
4.53. Admissible Paths for Elastic Behavior and Phase Transformations	195
4.54. Stress-Strain Behavior for Multilinear Elasticity	198
8.1. Evaluating Interpolation for Quadratic Data with a 6x6x6 Grid	314

List of Tables

3.1. Linear Material Properties (Defined via MP,Lab)	14
3.2. Supported Elements, Material Properties, Primary Variables	17
4.1. Porous Elasticity Model Constants	28
4.2. Implicit Creep Equations	82
4.3. Material Solution Parameter	129
4.4. Cam-clay Model Constants	135
4.5. Mohr-Coulomb Model Constants	143
4.6. Tension Rankine Yield Surface Constants (Optional)	143
4.7. Residual Strength Coupling Constants (Optional)	143
4.8. Plastic Potential (Optional)	143
4.9. Friction Angle Scaling Function (Optional)	143
4.10. Cohesion Scaling Function (Optional)	144
4.11. Tension Strength Scaling Function (Optional)	144
4.12. Dilatation Angle Scaling Function (Optional)	144
4.13. Jointed Rock Constants for the Base Material	150
4.14. Tension Yield Surface for the Base Material (Optional)	150
4.15. Residual Strength Coupling for the Base Material (Optional)	151
4.16. Jointed Rock Constants for the Joint	151
4.17. Tension Yield Surface for the Joint (Optional)	151
4.18. Orientation of the Joint Failure Plane (Optional)	151
4.19. Drucker-Prager Concrete Model Constants	156
4.20. Rankine Surface Constant	157
4.21. Drucker-Prager Dilatancy Parameters	157
4.22. Exponential HSD Constants	159
4.23. Steel Reinforcement HSD Constants	161
4.24. Fracture Energy HSD Constants	163
4.25. Linear HSD Constants	164
4.26. Jointed Rock Constants for the Joint	165
4.27. Tension Yield Surface for the Joint (Optional)	165
4.28. Orientation of the Joint Failure Plane (Optional)	166
4.29. Superelastic Option Constants	192
4.30. Shape Memory Effect Option Constants	196
4.31. Support for Rayleigh Material Damping Defined using MP and TB Commands	230
4.32. Comparison: Structural Damping Coefficient Options	232
6.1. Experimental Details for Case 1 and 2 Models and Blatz-Ko	261
6.2. Experimental Details for Case 3 Models	261
6.3. Hyperelastic Curve-Fitting Model Types	262
6.4. Viscoelastic Data Types and Abbreviations	268
6.5. Creep Data Types and Abbreviations	287
6.6. Creep Model and Data/Type Attribute	288
6.7. Creep Models and Abbreviations	289
7.1. Material Model Combination Possibilities	297
8.1. Predefined Field Variables and Supported Material Data Tables	301

Chapter 1: Introduction to Material Models

Material models (also called constitutive models), are the mathematical representation of a material's response to an applied load.

Typical model classes include the relationships between stress-strain, heat flux-temperature gradient, voltage-strain, and current-voltage, but also include more general behaviors such as friction and bonding, and response due to changes in the physical environment such as thermal expansion and swelling.

This reference provides information about material model behavior and application, including details about the load-response relationship and the necessary information required to use the material models in an analysis. The models are grouped based on the degrees of freedom that, directly or indirectly, give the loading function that serves as the input for the material model.

The following related introductory topics are available:

- [1.1. Material Models for Displacement Applications](#)
- [1.2. Material Models for Temperature Applications](#)
- [1.3. Material Models for Electromagnetic Applications](#)
- [1.4. Material Models for Coupled Applications](#)
- [1.5. Material Parameters](#)
- [1.6. How Material Properties Are Evaluated](#)

1.1. Material Models for Displacement Applications

For analyses that include displacement degrees of freedom, the input is a function of deformation such as strain or displacement, and the response is given force-like quantities such as stress or normal and tangential forces. The following general material types are available:

Type	Behavior	Application
Linear elastic	The response is the stresses that are directly proportional to the strains and the material will fully recover the original shape when unloaded. For isotropic materials, the relationship is given by Hooke's law and this relationship can be generalized to define anisotropic behavior.	Many metals are linear elastic at room temperature when the strains are small.
Plastic and elastic-plastic	The deformation of the material includes a permanent, or plastic, component that will not return to the original configuration if the load is removed and evolves in response to the deformation history. These materials also typically have an elastic behavior so that the	Plastic deformation is observed in many materials such as metals, alloys, soils, rocks, concrete, and ceramics.

Type	Behavior	Application
	combined deformation includes a part that is recoverable upon unloading.	
Hyperelastic	The behavior of these models is defined by a strain-energy potential, which is the energy stored in the material due to strain. The mathematical formulation is convenient for large-deformation analyses.	Hyperelastic models are often used for materials that undergo large elastic deformation, such as polymers and biological materials.
Rate effects and time dependency	This is a general behavior in which the response of the material depends on the rate of deformation, and therefore also the time. Examples include viscoelasticity, viscoplasticity, creep and damping.	Metal alloys that show significant creep deformation under elevated temperature, rate-dependent metal forming applications, polymers which typically get stiffer for increased deformation rate, and structures that damp out high frequency waves under dynamic loading.
Expansion and swelling	Materials often respond to changes in the physical environment and this response affects the structural behavior. Examples include thermal expansion in which changes in material volume depend on changes in temperature and swelling behaviors that depend on hygroscopic effects or neutron flux.	Radiation environments, bonded materials with thermal strain mismatch, and soils that absorb water.
Interaction	These models produce a response based on the interaction of structures.	Gasket and joint materials and also models of bonded and separating surfaces along interfaces or material cleavage.
Shape memory alloy	An elastic constitutive model with an internal phase transformation.	The phase transformation depends on the stress and temperature that cause an internal transformation strain.

1.2. Material Models for Temperature Applications

For analyses that include temperature as a degree of freedom, the material model for conduction gives a heat flux due to the gradient of temperature and also interaction between bodies due to radiative heat transfer that is dependent on surface temperature differences.

1.3. Material Models for Electromagnetic Applications

Material models for use in analyses with electromagnetic degrees of freedom include:

Type	Description
Magnetic	Gives the magnetomotive force in response to the magnetic flux.
Conductivity	For electric and magnetic current conductivities that model the relationship between the respective field and its flux.
Permittivity and Permeability	Gives the energy storage in a material in response to an electromagnetic field.
Loss	Represents the energy lost in electromagnetic and dielectric materials in response to changes in electromagnetic fields.

1.4. Material Models for Coupled Applications

Some models are valid in analyses that couple the thermal, electromagnetic, and displacement degrees of freedom.

Although the models are coupled, they remain distinct and give the same load-response behavior. However, the piezoelectric and piezoresistive materials are electromechanical coupled models that give a strain in response to a voltage and also produce a voltage in response to straining.

1.5. Material Parameters

Because a material model represents a mathematical relationship between response and load, it requires input parameters so that the model matches the material behavior.

In some cases, the parameters can be a function of physical field quantities such as temperature, frequency or time or interaction quantities such as normal pressure, relative distance, or relative velocity.

Matching the model to the actual behavior can be challenging; therefore, some built-in curve-fitting methods are available that use minimization to select a set of parameters that give a close fit to measured material behavior. The curve-fitting methods help you to select material parameters for creep, hyperelastic, viscoelastic, and some plastic models.

1.6. How Material Properties Are Evaluated

Material properties are evaluated at or near the centroid of the element or at each of the integration points, as follows:

- **Heat-transfer elements:** All properties are evaluated at the centroid (except for the specific heat or enthalpy, which is evaluated at the integration points).
- **Structural elements:** All properties are evaluated at the integration points.
- **All other elements:** All properties are evaluated at the centroid.

If the temperature of the centroid or integration point falls below or rises above the defined temperature range of tabular data, ANSYS assumes the defined extreme minimum or maximum value, respectively, for the material property outside the defined range.

Chapter 2: Material Model Element Support

Following is a list of available material models and the elements that support each material.

Material models are specified via the **TB,Lab** command, where *Lab* represents the material model label (shortcut name).

For a list of *elements* and the material models they support (*Lab* value), see [Element Support for Material Models in the Element Reference](#).

Label (<i>Lab</i>)	Material Model	Elements
AHYPER	Anisotropic hyperelasticity (p. 93)	SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263 (plane stress option), REINF265 (plane stress option), SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
ANEL	Anisotropic elasticity (p. 19)	SOLID5, PLANE13, SOLID98, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263 (plane stress option), REINF265 (plane stress option), SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
AVIS	Anisotropic viscosity (p. 247)	PLANE223, SOLID226, SOLID227
BB	Bergstrom-Boyce (p. 94)	PLANE182, PLANE183, SHELL181, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263 (plane stress option), REINF265 (plane stress option), SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
BH	Magnetic (p. 245)	SOLID5, PLANE13, SOLID96, SOLID98, PLANE223, PLANE233, SOLID226, SOLID227, SOLID236, SOLID237
BISO	Bilinear isotropic hardening (p. 40)	von Mises plasticity: LINK180, SHELL181, , PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, CABLE280, SHELL281, PIPE288, PIPE289, ELBOW290 Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188,

Label (<i>Lab</i>)	Material Model	Elements
		BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
BKIN	Bilinear kinematic hardening (p. 47)	von Mises plasticity: LINK180, SHELL181, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, CABLE280, SHELL281, PIPE288, PIPE289, ELBOW290, PLANE182, PLANE183 Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
CAST	Cast iron (p. 69)	PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, SOLID285, PIPE288, PIPE289
CDM	Mullins effect (p. 95)	SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263 (plane stress option), REINF265 (plane stress option), SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
CGCR	Crack growth fracture criterion	PLANE182, SOLID185
CHABOCHE	Chaboche nonlinear kinematic hardening (p. 51)	von Mises or Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290

Label (Lab)	Material Model	Elements
CONCR	Drucker-Prager concrete (p. 153) (TB ,CONCR,,,,DP)	PLANE182 (excluding plane stress), PLANE183 (excluding plane stress), SOLID185 (excluding layered structural solid), SOLID186 (excluding layered structural solid), SOLID187, CPT212, CPT213, CPT215, CPT216, CPT217
	Mentrey-Willam (p. 167) (TB ,CONCR,,,,MW)	
CREEP	Creep (p. 81)	Implicit creep with von Mises or Hill potential: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, SOLID285, CABLE280, SHELL281, PIPE288, PIPE289, ELBOW290
CTE	Coefficient of thermal expansion (p. 21)	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
CZM	Cohesive zone (p. 209)	CONTA172, CONTA174, CONTA175, CONTA177, INTER202, INTER203, INTER204, INTER205
DENS	Mass density	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, REINF263, REINF264, REINF265, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
DLST	Anisotropic dielectric loss tangent (p. 250)	PLANE223, SOLID226, SOLID227
DMGE	Damage evolution law (p. 221)	Progressive damage evolution (MPDG option): LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290 Continuum damage mechanics (CDM option): SHELL181, PLANE182 (plane stress option), PLANE183 (plane stress option), SHELL208, SHELL209, REINF263 (plane stress option), REINF265 (plane stress option), SHELL281, PIPE288 (thin pipe formulation), PIPE289 (thin pipe formulation), ELBOW290

Label (<i>Lab</i>)	Material Model	Elements
DMGI	Damage initiation criteria (p. 220)	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, REINF263, REINF264, REINF263, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
DPER	Anisotropic electric permittivity (p. 246)	PLANE223, SOLID226, SOLID227
EDP [1]	Extended Drucker-Prager (p. 55)	PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, SOLID285, PIPE288, PIPE289
ELASTIC	Elasticity	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290
ELST	Anisotropic elastic loss tangent (p. 249)	PLANE223, SOLID226, SOLID227
EXPE	Experimental data (p. 26)	<i>Used only with other material models</i>
FCON	Fluid conductance data	FLUID116
FCLI	Failure criteria material strength limits (p. 218)	All structural elements
FLUID	Fluid (p. 242)	HSFLD241, HSFLD242
FRIC	Coefficient of friction (p. 204)	CONTA172, CONTA174, CONTA175, CONTA177, CONTA178 Orthotropic friction (TB ,FRIC,,,,ORTHO) is not applicable to the 2-D contact element, CONTA172, or the node-to-node contact element, CONTA178.
GASKET	Gasket (p. 184)	INTER192, INTER193, INTER194, INTER195
GURSON	Gurson pressure-dependent plasticity (p. 64)	PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, SOLID285
HFLM	Film coefficient data	FLUID116
HILL	Hill anisotropy (p. 38)	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290

Label (Lab)	Material Model	Elements
HYPER	Hyperelasticity (p. 83)	SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, CPT212, CPT213, CPT215, CPT216, CPT217, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, REINF263 (plane stress option), REINF265 (plane stress option), SOLID227, SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290,
INTER	Contact interaction (p. 207)	User-defined contact interaction: CONTA172, CONTA174, CONTA175, CONTA177, CONTA178 Contact interactions for general contact: CONTA172, CONTA174, CONTA175, CONTA177
JOIN	Joint (p. 199) (linear and nonlinear elastic stiffness, linear and nonlinear damping, and frictional behavior)	MPC184
JROCK	Jointed rock (p. 145)	PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185 (not applicable for layered structural solid), SOLID186 (not applicable for layered structural solid), SOLID187, CPT212, CPT213, CPT215, CPT216, CPT217
MC	Mohr-Coulomb (p. 136)	PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185 (not applicable for layered structural solid), SOLID186 (not applicable for layered structural solid), SOLID187, CPT212, CPT213, CPT215, CPT216, CPT217
MELAS	Multilinear elasticity (p. 197)	PLANE182 (plane strain or axisymmetric), PLANE183 (plane strain or axisymmetric), SOLID185, SOLID186, SOLID187, SOLSH190, SOLID272, SOLID273, SOLID285, PIPE288, PIPE289
MIGR	Migration model (p. 251)	PLANE223, SOLID226, SOLID227
MPLANE	Microplane (p. 107)	PLANE182 and PLANE183 (plane stress excluded), SOLID185, SOLID186, SOLID187, CPT212, CPT213, CPT215, CPT216, CPT217
NLISO	Voce isotropic hardening law (p. 43)	von Mises or Hill plasticity: LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290

Label (<i>Lab</i>)	Material Model	Elements
PELAS	Porous elasticity (p. 26)	PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185 (not applicable for layered structural solid), SOLID186 (not applicable for layered structural solid), SOLID187, CPT212, CPT213, CPT215, CPT216, CPT217
PERF	Perforated Media (p. 233)	FLUID30, FLUID220, FLUID221
PIEZ	Piezoelectric matrix (p. 243)	SOLID5, PLANE13, SOLID98, PLANE223, SOLID226, SOLID227
PLASTIC	Plasticity (p. 28)	LINK180, SHELL181, PIPE288, PIPE289, ELBOW290, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, CPT212, CPT213, CPT215, CPT216, CPT217, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285
PM	Coupled Pore-Fluid Diffusion and Structural Model of Porous Media (p. 180)	CPT212, CPT213, CPT215, CPT216, CPT217
PRONY	Prony series (p. 96) constants for viscoelastic materials	LINK180, SHELL181, PIPE288, PIPE289, ELBOW290, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285
PZRS	Piezoresistivity (p. 244)	PLANE223, SOLID226, SOLID227
RATE	Rate-dependent plasticity (p. 81) (viscoplasticity)	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290 Anand unified plasticity option (p. 80): SHELL181 (except plane stress), PLANE182 (except plane stress), PLANE183 (except plane stress), SOLID185, SOLID186, SOLID187, SOLSH190, SOLID272, SOLID273, SOLID285, PIPE288, PIPE289
SDAMP (<i>TB-OPT</i> = STRU)	Material-Dependent Structural Damping (p. 230)	SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, SOLID272, SOLID273, SHELL281, SOLID285, ELBOW290

Label (Lab)	Material Model	Elements
SDAMP (TB- OPT = ALPD / BETD)	Material-Dependent Alpha and Beta Damping (Rayleigh Damping) (p. 227)	SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL208, SHELL209, SHELL281, SOLID285
SHIFT	Shift function (p. 96) for viscoelastic materials	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, PLANE222, PLANE223, SOLID226, SOLID227, PIPE288, PIPE289, ELBOW290, REINF263, REINF264, REINF265, SOLID272, SOLID273, CABLE280, SHELL281, SOLID285
SMA	Shape memory alloy (p. 188)	PLANE182, PLANE183, PLANE222, PLANE223 (with plane strain or axisymmetric stress states), SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, SOLID226, SOLID227, REINF263 (plane stress option), REINF265 (plane stress option), SOLID272, SOLID273, SHELL281, SOLID285, PIPE288, PIPE289
SOIL	Cam-clay (p. 131)	PLANE182 (not applicable for plane stress), PLANE183 (not applicable for plane stress), SOLID185 (not applicable for layered structural solid), SOLID186 (not applicable for layered structural solid), SOLID187, CPT212, CPT213, CPT215, CPT216, CPT217
STATE	State variables (user-defined) (p. 217)	FLUID116, LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, REINF263, REINF264 (structural and thermal), REINF265 (structural and thermal), SOLID272, SOLID273, SOLID278, SOLID279, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290, SOLID291, PLANE292, PLANE293 Also, user-defined plasticity or viscoplasticity: PLANE183
SWELL	Swelling (p. 186)	SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, SHELL281, SOLID285
THERM	Thermal properties (p. 257)	SOLID278, SOLID279, SOLID291, PLANE292, PLANE293, CPT212, CPT213, CPT215, CPT216, CPT217
USER	User-defined (UserMat) (p. 215)	LINK180, SHELL181, PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, BEAM188, BEAM189, SOLSH190, SHELL208, SHELL209, SOLID226, SOLID227, REINF263, REINF264, REINF265, SOLID272, SOLID273, SOLID278, SOLID279, SHELL281, SOLID285, PIPE288, PIPE289, ELBOW290, SOLID291, PLANE292, PLANE293 Also, user-defined plasticity or viscoplasticity: PLANE183
	User-defined thermal (UserMatTh) (p. 216)	FLUID116 (KEYOPT(1) = 1), SOLID226, SOLID227, REINF264, REINF265, SOLID278, SOLID279, SOLID291, PLANE292, PLANE293
WEAR	Contact surface wear (p. 214)	CONTA172, CONTA174, CONTA175

1. For this material model, the CPT nnn elements do not support the EDP cap yield function (**TB**,Lab,MAT,NTEMP,NPTS,TBOPT = CYFUN).

Chapter 3: Linear Material Properties

Material properties (which may be functions of temperature) are described as *linear* properties because most nonthermal analyses with those properties require only a single iteration. Material properties for a single-solution analysis are typically input via the **MP** command.

Note:

The **TB** command can also be used to input some linear material properties, such as [anisotropic elasticity](#) (p. 19), [material structural damping](#), and [piezoelectric matrix](#) (p. 243).

If properties needed for a thermal analysis (such as KXX) are temperature-dependent, the problem is considered to be *nonlinear*. Properties such as [stress-strain data](#) (p. 28) are nonlinear because an analysis with these properties requires an iterative solution.

Linear material properties required by an element but that remain undefined use default values. (Exceptions are EX and KXX, which must be input with a nonzero value where applicable.) The program ignores any additional material properties.

The X, Y, and Z portions of the material property labels refer to the element coordinate system. In general, [if a material is isotropic](#) (p. 18), only the X and possibly the XY term are input.

The following topics for linear material properties are available:

- [3.1. Defining Linear Material Properties](#)
- [3.2. Stress-Strain Relationships](#)
- [3.3. Anisotropic Elasticity](#)
- [3.4. Damping](#)
- [3.5. Thermal Expansion](#)
- [3.6. Emissivity](#)
- [3.7. Specific Heat](#)
- [3.8. Film Coefficients](#)
- [3.9. Temperature Dependency](#)

3.1. Defining Linear Material Properties

The linear material properties used by the element type are listed under **Material Properties** in the input table for each element type.

Tabular input (p. 16) is also available for some material properties.

Table 3.1: Linear Material Properties (Defined via **MP,Lab)**

Material Property (Lab Value)	Units	Description
EX	Force/Area	Elastic modulus, element x direction
EY		Elastic modulus, element y direction
EZ		Elastic modulus, element z direction
PRXY	None	Major Poisson's ratio, x-y plane
PRYZ		Major Poisson's ratio, y-z plane
PRXZ		Major Poisson's ratio, x-z plane
NUXY		Minor Poisson's ratio, x-y plane
NUYZ		Minor Poisson's ratio, y-z plane
NUXZ		Minor Poisson's ratio, x-z plane
GXY	Force/Area	Shear modulus, x-y plane
GYZ		Shear modulus, y-z plane
GXZ		Shear modulus, x-z plane
ALPX	Strain/Temp	Secant coefficient of thermal expansion, element x direction
ALPY		Secant coefficient of thermal expansion, element y direction
ALPZ		Secant coefficient of thermal expansion, element z direction
CTEX	Strain/Temp	Instantaneous coefficient of thermal expansion, element x direction
CTEY		Instantaneous coefficient of thermal expansion, element y direction
CTEZ		Instantaneous coefficient of thermal expansion, element z direction
THSX	Strain	Thermal strain, element x direction
THSY		Thermal strain, element y direction
THSZ		Thermal strain, element z direction
REFT	Temp	Reference temperature (as a property) (see also TREF)
MU	None	Coefficient of friction (or, for FLUID29 , boundary admittance)
ALPD	None	Mass matrix multiplier for damping (also see ALPHAD)
BETD	None	Stiffness matrix multiplier for damping (also see BETAD)
DMPR	None	Damping ratio
DMPS	None	Constant structural damping coefficient
DENS	Mass/Vol	Mass density
KXX	Heat*Length/ (Time*Area*Temp)	Thermal conductivity, element x direction
KYY		Thermal conductivity, element y direction
KZZ		Thermal conductivity, element z direction

Material Property (Lab Value)	Units	Description
C	Heat/Mass*Temp	Specific heat
ENTH	Heat/Vol	Enthalpy ($\int \text{DENS} * C \, d(\text{Temp})$)
HF	Heat / (Time*Area*Temp)	Convection (or film) coefficient
EMIS	None	Emissivity
QRATE	Heat/Time	Heat generation rate for thermal mass element MASS71
	None	Fraction of plastic work converted to heat (Taylor-Quinney coefficient) for coupled-field elements PLANE222 , PLANE223 , SOLID226 , and SOLID227
VISC	Force*Time/ Length ²	Viscosity
SONC	Length/Time	Sonic velocity (FLUID29 , FLUID30 , FLUID129 , and FLUID130 elements only)
MURX	None	Magnetic relative permeability, element x direction
MURY		Magnetic relative permeability, element y direction
MURZ		Magnetic relative permeability, element z direction
MGXX	Current/Length	Magnetic coercive force, element x direction
MGYY		Magnetic coercive force, element y direction
MGZZ		Magnetic coercive force, element z direction
RSVX	Resistance*Area/Length	Electrical resistivity, element x direction
RSVY		Electrical resistivity, element y direction
RSVZ		Electrical resistivity, element z direction
PERX	None	Electric relative permittivity, element x direction
PERY		Electric relative permittivity, element y direction
PERZ		Electric relative permittivity, element z direction
LSST	None	Dielectric loss tangent
SBKX	Voltage/Temp	Seebeck coefficient, element x direction
SBKY		Seebeck coefficient, element y direction
SBKZ		Seebeck coefficient, element z direction
DXX	Length ² /Time	Diffusion coefficient, element x direction
DYY		Diffusion coefficient, element y direction
DZZ		Diffusion coefficient, element z direction
CSAT	Mass/Length ³	Saturated concentration
CREF	Mass/Length ³	Reference concentration
BETX	Length ³ /Mass	Coefficient of diffusion expansion, element x direction
BETY		Coefficient of diffusion expansion, element y direction
BETZ		Coefficient of diffusion expansion, element z direction

3.1.1. Defining Linear Material Properties Using Tabular Input

Some element types support material properties defined as a function of primary variables, accomplished by using tabular input when defining the property.

Tabular input for linear material properties is typically defined via the **MP** family of commands, although the **TB** family of commands can be used for [some linear material properties](#) (p. 13).

To input a table, dimension the table array parameter (***DIM**) and identify the variables. Enclose the table name within % characters:

```
MP,Lab,,%tablename%
```

The possible primary variables for material tabular input are:

- Time (TIME)
- X location (X) in local or global coordinates
- Y location (Y) in local or global coordinates
- Z location (Z) in local or global coordinates
- Temperature (TEMP) load or degree of freedom
- Concentration (CONC) degree of freedom
- Pressure (PRESSURE) degree of freedom
- Computed fluid velocity (VELOCITY)

Example 3.1: Using Tabular Input (via **MP**) with One Primary Variable to Define Resistivity

A temperature-dependent resistivity is specified using a table array (as an alternative to using **MPTEMP** and **MPDATA**).

```
*DIM,RSV_T, TABLE,5,,TEMP      ! Define RSV_T with TEMP
                                !           primary variable
RSV_T(1,0)=0.,50.,100.,150.,200. ! Assign temperature values
RSV_T(1,1)=1.e-8,1.4e-8,1.6e-8,1.8e-8,2.e-8 ! Assign resistivity values

MP,RSVX,1,%RSV_T%              ! Input RSV_T on the MP command
```

Example 3.2: Using Tabular Input (via **MP**) with Two Primary Variables to Define the Diffusivity Coefficient

The diffusivity coefficient is dependent on both concentration (CONC) and time (TIME).

```
*DIM,DXX_CT, TABLE,3,4,,CONC,TIME ! Define DXX_CT with
                                !   CONC and TEMP primary variables
*TAXIS,DXX_CT,1, 0,1,2           ! X-axis is CONC, normalized
*TAXIS,DXX_CT,2, 1,2,3,4        ! Y-axis is TIME, hours
DXX_CT(1,1)= 2.e-6,3.e-6,4.e-6  ! Assign DXX vs CONC at TIME = 1 hr
DXX_CT(1,2)= 1.6e-6,2.4e-6,3.8e-6 ! Assign DXX vs CONC at TIME = 2 hr
DXX_CT(1,3)= 1.4e-6,2.2e-6,3.5e-6 ! Assign DXX vs CONC at TIME = 3 hr
DXX_CT(1,4)= 1.1e-6,2.e-6,3.1e-6 ! Assign DXX vs CONC at TIME = 4 hr

MP,DXX,1,%DXX_CT%              ! Input DXX_CT on the MP command
```

To input a table via the **TBDATA** command, dimension the table array parameter (***DIM**) and identify the variables. Enclose the table name within % signs.

TBDATA,STLOC,%tablename%

where *STLOC* is the starting location in the table for entering data. (Default = last location filled + 1.)

Example 3.3: Using Tabular Input (via **TBDATA**) with the TEMP Primary Variable

The temperature-dependent elastic coefficient c66 is defined for anisotropic elasticity (**TB,ANEL**).

```
*DIM, c66_T, TABLE, 5, , , TEMP
c66_T(1,0)= 20,40,60,80,100          ! Temperature
c66_T(1,1)= 39.8e9,40.0e9,40.1e9,40.3e9,40.4e9 ! Stiffness coefficient

TB,ANEL,1,,0          ! Anisotropic elasticity material data table
...
TBDATA,21,%c66_T%     ! Tabular input for elastic constant 21
```

Table 3.2: Supported Elements, Material Properties, Primary Variables

Material Property	Primary Variables
Coupled-Field Solids: PLANE223, SOLID226, SOLID227	
Diffusivity coefficients (DXX, DYY, DZZ)	TIME, X, Y, Z, TEMP, CONC
Saturated concentration (CSAT)	TIME, X, Y, Z
Electrical resistivities (RSVX, RSVY, RSVZ)	TIME, X, Y, Z, TEMP, CONC
Coefficients of diffusion expansion (BETX, BETY, BETZ)	TIME, X, Y, Z, TEMP, CONC
Reference concentration (CREF)	TIME, X, Y, Z, TEMP, CONC
Anisotropic Elasticity (ANEL)	TEMP [1]
Piezoelectricity (PIEZ)	TEMP [1]
Anisotropic Electric Permittivity (DPER)	TEMP [1]
Diffusion Solids: PLANE238, SOLID239, SOLID240	
Diffusivity coefficients (DXX, DYY, DZZ)	TIME, X, Y, Z, TEMP, CONC
Saturated concentration (CSAT)	TIME, X, Y, Z
Coupled Thermal-Fluid Pipe: FLUID116	
Thermal conductivity (KXX)	TIME, X, Y, Z, TEMP, PRESSURE, VELOCITY
Specific heat (C)	TIME, X, Y, Z, TEMP, PRESSURE, VELOCITY
Mass Density (DENS)	TIME, X, Y, Z, TEMP, PRESSURE, VELOCITY
Coefficient of friction (MU)	TIME, X, Y, Z, TEMP, PRESSURE, VELOCITY
Viscosity (VISC)	TIME, X, Y, Z, TEMP, PRESSURE, VELOCITY
Film coefficient (HF)	TIME, X, Y, Z, TEMP, PRESSURE, VELOCITY
Hydrodynamic Bearing: FLUID218	

Material Property	Primary Variables
Viscosity (VISC)	X, Y, Z, TEMP, PRESSURE
Mass density (DENS)	X, Y, Z, TEMP, PRESSURE
Thermal Mass: MASS71	
Heat-generation rate (QRATE)	TIME, X, Y, Z, TEMP

1. **TBDATA** required.

For more information about using tabular input and additional examples, see:

- [Applying Loads Using Tabular Input in the *Basic Analysis Guide*](#)
- [Tabular Input via Table Array Parameters in the *ANSYS Parametric Design Language Guide*](#)
- [Using Tabular Input to Define Matrix Coefficients in the *Coupled-Field Analysis Guide*](#)
- ***DIM**

Information about element-specific usage of tabular input for material properties is available in the description of the element that you are using.

3.2. Stress-Strain Relationships

Structural material properties must be input as an isotropic, orthotropic, or anisotropic material.

If the material is isotropic:

- Young's modulus (EX) must be input.
- Poisson's ratio (PRXY or NUXY) defaults to 0.3. If a zero value is desired, input PRXY or NUXY with a zero or blank value. Poisson's ratio should not be ≥ 0.5 nor ≤ -1.0 .
- The shear modulus (GXY) defaults to $EX/(2(1+NUXY))$. If GXY is input, it must match $EX/(2(1+NUXY))$. The GXY value ensures consistency with the other two properties.

If the material is orthotropic:

- EX, EY, EZ, (PRXY, PRYZ, PRXZ, or NUXY, NUYZ, NUXZ), GXY, GYZ, and GXZ must all be input if the element type uses the material property. There are no defaults. For example, if only EX and EY are input (with different values) to a plane stress element. The program generates an error message indicating that the material is orthotropic and that GXY and NUXY are also needed.
- Poisson's ratio may be input in either major (PRXY, PRYZ, PRXZ) or minor (NUXY, NUYZ, NUXZ) form, but not both for a particular material. The major form is converted to the minor form during the solve operation (**SOLVE**). Solution output is in terms of the minor form, regardless of how the data was input. If zero values are desired, input the labels with a zero (or blank) value.
- For axisymmetric analyses, the X, Y, and Z labels refer to the radial (R), axial (Z), and hoop (θ) directions, respectively. Orthotropic properties given in the R, Z, θ system should be input as follows: EX = ER, EY = EZ, and EZ = E θ . An additional transformation is required for Poisson's ratios. If the given R, Z, θ properties are column-normalized (see the [Mechanical APDL Theory Reference](#)), NUXY = NURZ, NUYZ =

$NUZ \theta = (ET/EZ) * NU \theta Z$, and $NUXZ = NUR \theta$. If the given R, Z, θ properties are row-normalized, $NUXY = (EZ/ER)*NURZ$, $NUYZ = (E \theta/EZ)*NUZ \theta = NU \theta Z$, and $NUXZ = (E \theta/ER)*NUR \theta$.

- For all other orthotropic material properties (including ALPX, ALPY, and ALPZ), the X, Y, and Z part of the label (as in KXX, KYY, and KZZ) refers to the direction (in the element coordinate system) in which that particular property acts. The Y and Z directions of the properties default to the X direction (for example, KYY and KZZ default to KXX) to reduce the amount of input required.

If the material is anisotropic:

- See [Anisotropic Elasticity \(p. 19\)](#).

3.3. Anisotropic Elasticity

Anisotropic elastic capability (**TB**,ANEL or **TB**,ELAS,,,,AELS/AELF) is available with current-technology plane and solid elements. Input the elastic coefficient matrix [D] either by specifying the stiffness constants (EX, EY, etc.) with **MP** commands, or by specifying the terms of the matrix via data-table commands (**TB**). The matrix should be symmetric and positive definite (requiring all determinants to be positive).

The full 6 x 6 elastic coefficient matrix [D] relates terms ordered x, y, z, xy, yz, xz via 21 constants as shown below.

$$\begin{bmatrix} D_{11} & & & & & \\ D_{21} & D_{22} & & & & \\ D_{31} & D_{32} & D_{33} & \text{Symmetric} & & \\ D_{41} & D_{42} & D_{43} & D_{44} & & \\ D_{51} & D_{52} & D_{53} & D_{54} & D_{55} & \\ D_{61} & D_{62} & D_{63} & D_{64} & D_{65} & D_{66} \end{bmatrix}$$

For 2-D problems, a 4 x 4 matrix relates terms ordered x, y, z, xy via 10 constants ($D_{11}, D_{21}, D_{22}, D_{31}, D_{32}, D_{33}, D_{41}, D_{42}, D_{43}, D_{44}$). The order of the vector is expected as {x, y, z, xy, yz, xz}, whereas for some published materials the order is given as {x, y, z, yz, xz, xy}. This difference requires the "D" matrix terms to be converted to the expected format. The "D" matrix can be defined in either "stiffness" form (with units of Force/Area operating on the strain vector) or in "compliance" form (with units of the inverse of Force/Area operating on the stress vector), whichever is more convenient. Both forms use the same data-table input.

Define the elastic coefficient matrix via the **TB** family of commands:

1. Initialize the constant table (**TB**,ANEL or **TB**,ELAS,,,,AELS/AELF). Select the stiffness or flexibility form via the appropriate **TBOPT** value.
2. Define the temperature (**TBTEMP**), or issue the **TBFIELD** command with **TB**,ELAS to input temperature or [other supported field variables \(p. 301\)](#).
3. Specify up to 21 constants (**TBDATA**).

Constant	Meaning
C1-C6	Terms $D_{11}, D_{21}, D_{31}, D_{41}, D_{51}, D_{61}$
C7-C12	Terms $D_{22}, D_{32}, D_{42}, D_{52}, D_{62}, D_{33}$

Constant	Meaning
C13-C18	Terms D_{43} , D_{53} , D_{63} , D_{44} , D_{54} , D_{64}
C19-C21	Terms D_{55} , D_{65} , D_{66}

For **TB,ANEL**, you can define up to six temperature-dependent sets of constants (via *NTEMP* = 1 through 6). Matrix terms are linearly interpolated between temperature points.

Temperature dependence via *NTEMP* on the **TB,ANEL** command is not supported for coupled-field analyses.

In a [piezoelectric analysis](#) using coupled-field elements **PLANE223**, **SOLID226**, or **SOLID227**, you can define temperature-dependent elasticity coefficients via **TB,ANEL** (with *TBOPT* = 0) and tabular input on the **TB,DATA** command:

```
TB,DATA,STLOC,%tabname%
```

where *tabname* is the name of a table array parameter (***DIM**) with TEMP as a primary variable. For more information, see [Defining Linear Material Properties Using Tabular Input](#) (p. 16).

For a list of the elements that support this material model, see [Material Model Element Support](#) (p. 5).

3.4. Damping

Material-dependent mass and stiffness damping (**MP,ALPD** and **MP,BETD**) is an additional method of including damping for dynamic analyses and is useful when different parts of the model have different damping values.

ALPD and BETD are not assumed to be temperature dependent and are always evaluated at $T = 0.0$. Special-purpose elements, such as **MATRIX27** and **FLUID29**, generally do not require damping. However, if material property ALPD and BETD are specified for these elements, the value will be used to create the damping matrix at solution time.

Constant material damping coefficient (**MP,DMPS**) is a material-dependent structural damping coefficient that is constant with respect to the excitation frequency in harmonic analysis and is useful when different parts of the model have different damping values (see [Damping Matrices](#) in the *Mechanical APDL Theory Reference*). DMPS is not temperature-dependent and is always evaluated at $T = 0.0$.

For composite elements (such as SHELL, REINF, and Layered SOLID elements), the **MAT** command specifies a single material (the element material) and the **SECDATA** command defines additional composite materials. The program considers only element material damping (**MP,ALPD**, **MP,BETD**, or **MP,DMPS**) defined with the element material and applies it to the element as a whole (and ignores element material damping defined with additional composite materials).

If you require different Rayleigh damping coefficients for different composite materials, use the corresponding data-table commands (**TB,SDAMP,,,ALPD**, **TB,SDAMP,,,BETD**). For more information, see [Material Damping](#) (p. 227).

See [Damping Matrices](#) in the *Mechanical APDL Theory Reference* for more details about the damping formulation.

See [Damping](#) in the *Structural Analysis Guide* for more information.

3.5. Thermal Expansion

The uniform temperature does not default to REFT (but does default to TREF on the **TREF** command).

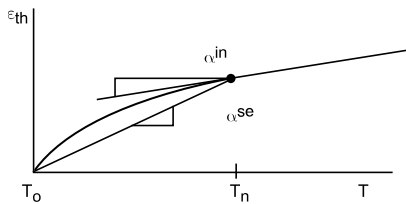
The effects of thermal expansion can be accounted for in one of three different ways:

- Secant coefficient of thermal expansion (ALPX, ALPY, ALPZ via the **MP** or **TB,CTE** command)
- Instantaneous coefficient of thermal expansion (CTEX, CTEY, CTEZ via the **MP** command)
- Thermal strain (THSX, THSY, THSZ via the **MP** command)

When you use ALPX to enter values for the secant coefficient of thermal expansion (α^{se}), the program interprets those values as secant or mean values, taken with respect to some common datum or *definition* temperature.

For example, suppose you measured thermal strains in a test laboratory, starting at 23°C, and took readings at 200°, 400°, 600°, 800°, and 1000°. When you plot this strain-temperature data, you could input this directly via THSX. The slopes of the secants to the strain-temperature curve would be the mean (or secant) values of the coefficient of thermal expansion, defined with respect to the common temperature of 23° (T_0).

You can also input the *instantaneous* coefficient of thermal expansion (α^{in} , using CTEX). The slopes of the tangents to this curve represent the instantaneous values. The following figure shows how alternate ways of inputting coefficients of thermal expansion relate to each other:



The program calculates structural thermal strain as follows:

$$\epsilon^{th} = \alpha^{se}(T) * (T - TREF)$$

where:

T = element evaluation temperature

TREF = temperature at which zero thermal strains exist (**TREF** command or REFT)

$\alpha^{se}(T)$ = secant coefficient of thermal expansion, with respect to a definition temperature (in this case, same as TREF) (ALPX)

If the material property data is in terms of instantaneous values of α , then the program will convert those instantaneous values into secant values as follows:

$$\alpha^{se}(T_n) = \frac{\int_{T_0}^{T_n} \alpha^{in}(T) * dT}{(T_n - TREF)}$$

where:

T_n = temperature at which an α^{se} value is being evaluated

T_o = definition temperature at which the α^{se} values are defined (in this case, same as TREF)

$\alpha^{in}(T)$ = instantaneous coefficient of thermal expansion at temperature T (CTEX)

If the material property data is in terms of thermal strain, the program will convert those strains into secant values of coefficients of thermal expansion as follows:

$$\alpha^{se}(T_n) = \frac{\epsilon^{ith}(T)}{(T_n - T_{ref})}$$

where:

$\epsilon^{ith}(T)$ = thermal strain at temperature T (THSX)

If necessary, the data is shifted so that the thermal strain is zero when $T_n = T_{ref}$. If a data point at T_{ref} exists, a discontinuity in α^{se} may be generated at $T_n = T_{ref}$. This can be avoided by ensuring that the slopes of ϵ^{ith} on both sides of T_{ref} match.

If the α^{se} values are based upon a definition temperature other than TREF, it is necessary to convert those values to TREF (**MPAMOD**).

Thermal expansion is assumed to be isotropic when used with any [hyperelasticity \(p. 83\)](#) material model. The program uses the *first* secant coefficient (ALPX), instantaneous coefficient (CTEX), or thermal strain coefficient (THSX).

The secant thermal expansion is assumed to be isotropic if only the first secant coefficient (ALPX for **MP**, or CTEX for **TB,CTE**) is specified.

3.6. Emissivity

EMIS defaults to 1.0 if not defined. However, if EMIS is set to zero or blank, EMIS is taken to be 0.0.

3.7. Specific Heat

You can input specific heat effects using either the C (specific heat) or ENTH (enthalpy) property. Enthalpy has units of heat/volume and is the integral of C x DENS over temperature. If both C and ENTH are specified, the program uses ENTH. ENTH should be used only in a transient thermal analysis. For phase-change problems, you must input ENTH as a function of temperature using the **MP** family of commands (**MP**, **MPTMP**, **MPTGEN**, and **MPDATA**).

3.8. Film Coefficients

Film coefficients are evaluated as described via the **SF** command. See the [Mechanical APDL Theory Reference](#) for additional details. Property evaluation at element temperatures beyond the supplied tabular

range assumes a constant property at the extreme range value. An exception occurs for the ENTH property, which continues along the last supplied slope.

3.9. Temperature Dependency

Temperature-dependent properties may be input in tabular form (value vs. temperature [**MPDATA** and **MPTEMP**]) or as a fourth-order polynomial (value = $f(\text{temperature})$ [**MP**]). If input as a polynomial, the program evaluates the dependencies at discrete temperature points during PREP7 preprocessing and then converts the properties to tabular form. The tabular properties are then available to the elements.

Chapter 4: Nonlinear Material Properties

Material properties are described as *nonlinear* because analyses using those properties require multiple solution iterations. Nonlinear material properties require [data-table input \(p. 26\)](#), defined via the **TB** command.

- 4.1. Understanding Material Data Tables
- 4.2. Experimental Data
- 4.3. Porous Elasticity
- 4.4. Rate-Independent Plasticity
- 4.5. Rate-Dependent Plasticity (Viscoplasticity)
- 4.6. Hyperelasticity
- 4.7. Viscoelasticity
- 4.8. Microplane
- 4.9. Geomechanics
- 4.10. Porous Media
- 4.11. Gasket
- 4.12. Swelling
- 4.13. Shape Memory Alloy (SMA)
- 4.14. Multilinear Elasticity
- 4.15. MPC184 Joint
- 4.16. Contact Friction
- 4.17. Contact Interaction
- 4.18. Cohesive Material Law
- 4.19. Contact Surface Wear
- 4.20. Custom Material Models
- 4.21. Material Strength Limits
- 4.22. Material Damage
- 4.23. Material Damping

For related information, see [Nonlinear Structural Analysis in the *Structural Analysis Guide*](#).

For information about multiphysics material models (which can have linear or nonlinear material properties), see [Multiphysics Material Properties \(p. 233\)](#).

For a list of the elements that support each material model, see [Material Model Element Support \(p. 5\)](#).

4.1. Understanding Material Data Tables

A material *data table* is a series of constants that are interpreted when they are used. Data tables are always associated with a material number and are most often used to define *nonlinear* material data (therefore requiring an iterative-solution analysis). Examples of nonlinear material data include stress-strain curves, creep constants, swelling constants, and magnetization curves. The material data table is defined via the **TB** command.

For some element types, the data table is used for special element input data other than material properties. The form of the data table depends upon the data being defined:

- Where the form is peculiar to only one element type, the table is described with the element in [Element Library](#).
- If the form applies to more than one element, it is described here and referenced in the element description.

4.2. Experimental Data

The experimental data-table option (**TB,EXPE**) allows you to input experimental data. The data is used with other material models.

Initiate the experimental data table, then specify the appropriate experimental data type (**TBOPT**), as follows:

UNIAXIAL - Uniaxial experimental data
BIAXIAL - Equibiaxial experimental data
SHEAR - Pure shear experimental data, also known as planar tension
VOLUMETRIC - Volumetric experimental data
SSHEAR - Simple shear experimental data
UNITENSION - Uniaxial tension experimental data
UNICOMPRESSION - Uniaxial compression experimental data
GMODULUS - Shear modulus experimental data
KMODULUS - Bulk modulus experimental data
EMODULUS - Tensile modulus experimental data
NUXY - Poisson's ratio experimental data

Enter the data (**TBPT**) for each data point. Each data point entered consists of the independent variable followed by one or more dependent variables. The specific definition of the input points depends on the requirements of the material model associated with the experimental data.

You can also define experimental data as a function of field variables. Field-dependent data are entered by preceding a set of experimental data (**TBFIELD**) to define the value of the field.

4.3. Porous Elasticity

Porous materials such as soils or polymer foams display nonlinear elastic behavior caused by the effect of voids on the bulk modulus of the material during hydrostatic compression.

Compared to the solid portion of the material, voids are relatively compressible, and the effect on the bulk modulus varies according to void proportions. During hydrostatic loading, voids compress or dilate; however, the solid portion of the material remains comparatively stiff, exhibiting little volumetric deformation. As the void ratio of the material changes, the bulk modulus also changes.

The rate form of the elastic stress-strain relationship is:

$$d\boldsymbol{\sigma}^{el} = \mathbf{C} d\boldsymbol{\varepsilon}^{el}$$

where $\mathbf{C}$ is the linearized elastic stiffness tensor, which can be given as a function of the bulk and shear moduli:

$$\mathbf{C} = k_b \mathbf{I} \otimes \mathbf{I} + 2\mu \mathbf{I}_{dev}$$

where k_b is the bulk modulus, μ is the shear modulus, $\mathbf{I}$ is the second-order identity tensor, and $\mathbf{I}_{dev}$ is the fourth-order deviatoric projection tensor.

For elastic loading, the change in void ratio is proportional to the change in logarithmic pressure:

$$de^{el} = -\kappa d(\ln p)$$

where κ is the swell index, e^{el} is the elastic void ratio and p is the pressure. If the material has a non-zero elastic tensile strength, the relationship is:

$$de^{el} = -\kappa d(\ln(p + p_t^{el})) \quad (4.1)$$

The elastic volumetric strain is related to the elastic void ratio:

$$\varepsilon_{vol}^{el} = \ln(J^{el}) = \ln\left(\frac{1+e^{el}}{1+e_0}\right) \quad (4.2)$$

where J^{el} is the elastic volume ratio. Integrating the porous elasticity relationship from [Equation 4.1 \(p. 27\)](#), substituting into [Equation 4.2 \(p. 27\)](#) and rearranging gives the pressure as a function of the elastic volumetric strain:

$$p = -p_t^{el} + \left(p_0 + p_t^{el}\right) \exp\left(\frac{1+e_0}{\kappa} \left(1 - \exp(\varepsilon_{vol}^{el})\right)\right) \quad (4.3)$$

where p_0 is the initial pressure. If the initial pressure is zero, then the model requires a non-zero elastic tensile strength.

The bulk modulus depends on the pressure and the elastic volumetric strain:

$$k_b = \left(p + p_t^{el}\right) \left(\frac{1+e_0}{\kappa} \exp \varepsilon_{vol}^{el}\right)$$

Caution:

Ensure small volumetric strain increments to prevent significant errors in the bulk modulus value used in the incremental stress update. This safeguard can be especially important when defining an initial stress that causes large volumetric deformation in the first solution substep.

The shear modulus does not depend on the void ratio or pressure directly, but Poisson's ratio is constant, resulting in the following relationship for the shear modulus:

$$\mu = \frac{3k_p(1-2\nu)}{2(1+\nu)}$$

where ν is the Poisson's ratio.

4.3.1. Defining the Porous Elasticity Model

To define the porous elasticity model:

1. Define the material data table (**TB,PELAS,,,,POISSON**).
2. Input the appropriate constants (**TBDATA**).
3. Define the initial stress state (p_0) (**INISTATE**).

Table 4.1: Porous Elasticity Model Constants

Constant	Meaning	Property
C1	κ	Swell index
C2	p_t^{el}	Elastic limit of tensile strength
C3	ν	Poisson's ratio
C4	e_0	Initial void ratio

Example 4.1: Defining the Porous Elasticity Model

```

/prep7
! Porous elasticity
Kappa = 0.0024
NU0    = 0.279
pt_el  = 5.835
E0     = 0.34
p0     = 69

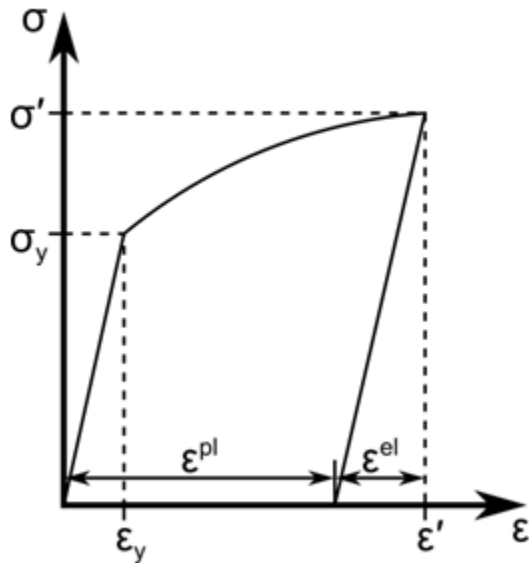
TB,PELAS,1,,,POISSON
TBDATA,1, Kappa, pt_el, NU0, E0

/solu
!define initial stress state
INISTATE,set,dtyp,stre
INISTATE,defi,all,,,,-p0,-p0,-p0,0,0,0

```

4.4. Rate-Independent Plasticity

Plasticity is used to model materials subjected to loading beyond their elastic limit. As shown in the following figure, metals and other materials such as soils often have an initial elastic region in which the deformation is proportional to the load, but beyond the elastic limit a nonrecoverable plastic strain develops:

Figure 4.1: Stress-Strain Curve for an Elastic-Plastic Material

Unloading recovers the elastic portion of the total strain, and if the load is completely removed, a permanent deformation due to the plastic strain remains in the material. Evolution of the plastic strain depends on the load history such as temperature, stress, and strain rate, as well as internal variables such as yield strength, backstress, and damage.

To simulate elastic-plastic material behavior, several constitutive models for plasticity are provided. The models range from simple to complex. The choice of constitutive model generally depends on the experimental data available to fit the material constants.

The following rate-independent plasticity material model topics are available:

- 4.4.1. Understanding the Plasticity Models
- 4.4.2. Isotropic Hardening
- 4.4.3. Kinematic Hardening
- 4.4.4. Drucker-Prager
- 4.4.5. Gurson
- 4.4.6. Cast Iron
- 4.4.7. Crushable Foam

4.4.1. Understanding the Plasticity Models

The constitutive models for elastic-plastic behavior start with a decomposition of the total strain into elastic and plastic parts and separate constitutive models are used for each. The essential characteristics of the plastic constitutive models are:

- The yield criterion that defines the material state at the transition from elastic to elastic-plastic behavior.
- The flow rule that determines the increment in plastic strain from the increment in load.
- The hardening rule that gives the evolution in the yield criterion during plastic deformation.

The following topics concerning plasticity theory, behavior and model definition are available:

4.4.1.1. Nomenclature

4.4.1.2. Strain Decomposition

4.4.1.3. Yield Criterion

4.4.1.4. Flow Rule

4.4.1.5. Hardening

4.4.1.6. Large Deformation

4.4.1.7. Output

4.4.1.8. Resources

4.4.1.1. Nomenclature

Following are the common symbols used in the rate-independent plasticity theory documentation:

Symbol	Definition	Symbol	Definition
$\mathbf{I}$	Identity tensor	σ_{ij}^y	Anisotropic directional yield strength
$\boldsymbol{\varepsilon}$	Strain	E	Young's Modulus
$\boldsymbol{\varepsilon}^{el}$	Elastic strain	E_T	Elasto-plastic tangent
$\boldsymbol{\varepsilon}^{pl}$	Plastic strain	E_{Ti}	Elasto-plastic tangent in direction i
ε_{ij}^{pl}	Plastic strain components	E^{pl}	Plastic tangent
$\bar{\varepsilon}^{pl}$	Effective plastic strain	E_i^{pl}	Plastic tangent in direction i
$\hat{\varepsilon}^{pl}$	Accumulated equivalent plastic strain	F, G, H, L, M, N	Hill yield surface coefficients
$\boldsymbol{\sigma}$	Stress	R_{ij}	Hill yield surface directional yield ratio
σ_{ij}	Stress components	<i>Reserved</i>	<i>Reserved</i>
σ_i	Principal stresses	<i>Reserved</i>	<i>Reserved</i>
$\bar{\sigma}$	Stress minus backstress	<i>Reserved</i>	<i>Reserved</i>
σ_y	Yield stress	κ	Plastic work
σ_i^y	Anisotropic yield stress in direction i	κ_x	Uniaxial plastic work
σ_0	Initial yield stress	β	Drucker-Prager yield surface constant
σ_{0i}	Initial yield stress in direction i	$\bar{\beta}$	Drucker-Prager plastic potential constant
$\hat{\sigma}^{pl}$	Equivalent plastic stress	c	Mohr-Coulomb cohesion
σ_e	Von Mises effective stress	ϕ	Mohr-Coulomb internal friction angle
$(\varepsilon_i, \sigma_i)$	User input strain-stress data point	ϕ_f	Mohr-Coulomb flow internal friction angle
$d\lambda$	Magnitude of plastic strain increment	α	Extended Drucker-Prager yield surface pressure sensitivity
$F(\boldsymbol{\sigma})$	Effective stress function	$\bar{\alpha}$	Extended Drucker-Prager plastic potential pressure sensitivity

Symbol	Definition	Symbol	Definition
f	Yield function	b	Extended Drucker-Prager power law yield exponent
Q	Plastic potential	$\bar{b}$	Extended Drucker-Prager power law plastic potential exponent
α	Translation of yield surface (backstress)	a	Extended Drucker-Prager hyperbolic yield constant
ξ	Set of material internal variables	$\bar{a}$	Extended Drucker-Prager hypobolic plastic potential constant

4.4.1.2. Strain Decomposition

From [Figure 4.1: Stress-Strain Curve for an Elastic-Plastic Material \(p. 29\)](#), a monotonic loading to σ' gives a total strain ε' . The total strain is additively decomposed into elastic and plastic parts:

$$\varepsilon' = \varepsilon^{el} + \varepsilon^{pl}$$

The stress is proportional to the elastic strain ε^{el} :

$$\sigma = D\varepsilon^{el}$$

and the evolution of plastic strain ε^{pl} is a result of the plasticity model.

For a general model of plasticity that includes arbitrary load paths, the flow theory of plasticity decomposes the incremental strain tensor into elastic and plastic strain increments:

$$d\varepsilon = d\varepsilon^{el} + d\varepsilon^{pl}$$

The increment in stress is then proportional to the increment in elastic strain, and the plastic constitutive model gives the incremental plastic strain as a function of the material state and load increment.

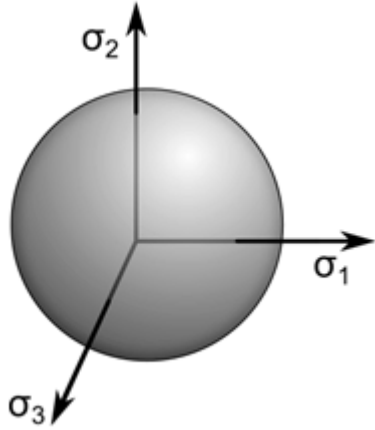
4.4.1.3. Yield Criterion

The yield criterion is a scalar function of the stress and internal variables and is given by the general function:

$$f(\sigma, \xi) = 0 \quad (4.4)$$

where ξ represents a set of history dependent scalar and tensor internal variables.

[Equation 4.4 \(p. 31\)](#) is a general function representing the specific form of the yield criterion for each of the plasticity models. The function is a surface in stress space and an example, plotted in principal stress space, as shown in this figure:

Figure 4.2: Yield Surface in Principal Stress Space

Stress states inside the yield surface are given by $f(\sigma, \xi) < 0$ and result in elastic deformation. The material yields when the stress state reaches the yield surface and further loading causes plastic deformation. Stresses outside the yield surface do not exist and the plastic strain and shape of the yield surface evolve to maintain stresses either inside or on the yield surface.

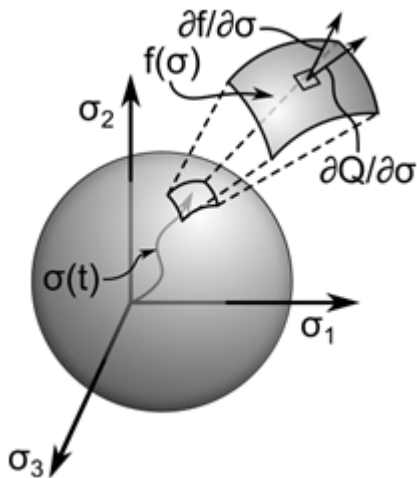
4.4.1.4. Flow Rule

The evolution of plastic strain is determined by the flow rule:

$$d\epsilon^{pl} = d\lambda \frac{\partial Q}{\partial \sigma}$$

where $d\lambda$ is the magnitude of the plastic strain increment and Q is the plastic potential.

When the plastic potential is the yield surface from [Equation 4.4 \(p. 31\)](#), the plastic strain increment is normal to the yield surface and the model has an associated flow rule, as shown in this figure:

Figure 4.3: Plastic Strain Flow Rule

These flow rules are typically used to model metals and give a plastic strain increment that is proportional to the stress increment. If the plastic potential is not proportional to the yield surface, the model has a non-associated flow rule, typically used to model soils and granular materials that plastically deform due to internal frictional sliding. For non-associated flow rules, the plastic strain increment is not in the same direction as the stress increment.

Non-associated flow rules result in an unsymmetric material stiffness tensor. Unsymmetric analysis can be set via the **NROPT** command. For a plastic potential that is similar to the yield surface, the plastic strain direction is not significantly different from the yield surface normal, and the degree of asymmetry in the material stiffness is small. In this case, a symmetric analysis can be used, and a symmetric material stiffness tensor will be formed without significantly affecting the convergence of the solution.

4.4.1.5. Hardening

The yield criterion for many materials depends on the history of loading and evolution of plastic strain. The change in the yield criterion due to loading is called *hardening* and is defined by the hardening rule. Hardening behavior results in an increase in yield stress upon further loading from a state on the yield surface so that for a plastically deforming material, an increase in stress is accompanied by an increase in plastic strain.

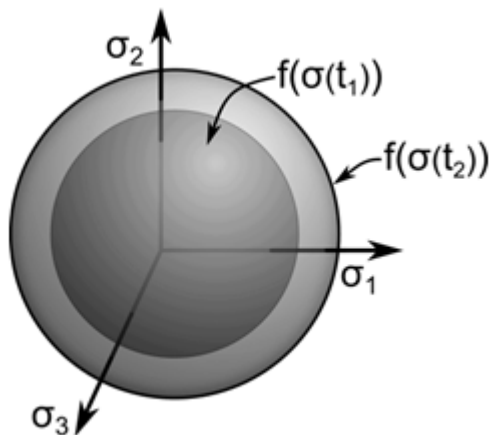
Two common types of hardening rules are isotropic and kinematic hardening. For isotropic hardening, the yield surface given by Equation 4.4 (p. 31) has the form:

$$F(\sigma) - \sigma_y(\xi) = 0$$

where $F(\sigma)$ is a scalar function of stress and $\sigma_y(\xi)$ is the yield stress.

Plastic loading from $\sigma(t_1)$ to $\sigma(t_2)$ increases the yield stress and results in uniform increase in the size of the yield surface, as shown in this figure:

Figure 4.4: Isotropic Hardening of the Yield Surface



This type of hardening can model the behavior of materials under monotonic loading and elastic unloading, but often does not give good results for structures that experience plastic deformation after a load reversal from a plastic state.

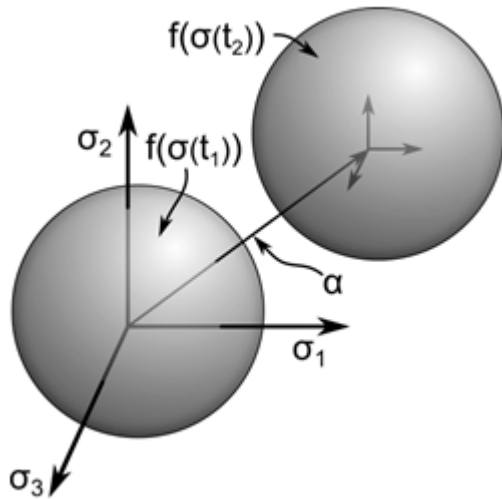
For kinematic hardening, the yield surface has the form:

$$f(\sigma - \alpha, \xi) = 0$$

where α is the backstress tensor.

As shown in the following figure, the backstress tensor is the center (or origin) of the yield surface, and plastic loading from $\sigma(t_1)$ to $\sigma(t_2)$ results in a change in the backstress and therefore a shift in the yield surface:

Figure 4.5: Kinematic Hardening of the Yield Surface



Kinematic hardening is observed in cyclic loading of metals. It can be used to model behavior such as the Bauschinger effect, where the compressive yield strength reduces in response to tensile yielding. It can also be used to model plastic ratcheting, which is the buildup of plastic strain during cyclic loading.

Many materials exhibit both isotropic and kinematic hardening behavior, and these hardening rules can be used together to give the combined hardening model. Other hardening behaviors include changes in the shape of the yield surface in which the hardening rule affects only a local region of the yield surface, and softening behavior in which the yield stress decreases with plastic loading.

4.4.1.6. Large Deformation

The plasticity constitutive models are applicable in both small-deformation and large-deformation analyses. For small deformation, the formulation uses engineering stress and strain. For large deformation (**NLGEOM,ON**), the constitutive models are formulated with the Cauchy stress and logarithmic strain.

Input quantities (such as the stress versus strain points for the multilinear isotropic hardening model) and output quantities (such as stress and elastic strain) use the stress and strain measures that correspond to the nonlinear geometry setting (**NLGEOM**).

4.4.1.7. Output

Output quantities specific to the plastic constitutive models are available for use in the **POST1** database postprocessor (**/POST1**) and in the **POST26** time-history results postprocessor (**/POST26**).

The equivalent stress (label SEPL) is the current value of the yield stress evaluated from the hardening model.

The accumulated plastic strain (label EPEQ) is a path-dependent summation of the plastic strain rate over the history of the deformation:

$$\hat{\varepsilon}^{pl} = \int d\lambda'$$

$$d\lambda' = \sqrt{\frac{2}{3} d\varepsilon_{ij}^{pl} d\varepsilon_{ij}^{pl}}$$

where $d\lambda'$ is the magnitude of the plastic strain increment.

The stress ratio (label SRAT) is the ratio of the elastic trial stress to the current yield stress and is an indicator of plastic deformation during an increment. If the stress ratio is:

> 1

A plastic deformation occurred during the increment.

< 1

An elastic deformation occurred during the increment.

1

The stress state is on the yield surface.

4.4.1.8. Resources

The following list of resources offers more information about plasticity:

1. Hill, R. *The Mathematical Theory of Plasticity*. New York: Oxford University Press, 1983.
2. Prager, W. "The Theory of Plasticity: A Survey of Recent Achievements." *Proceedings of the Institution of Mechanical Engineers*. 169.1 (1955): 41-57.
3. Besseling, J. F. "A Theory of Elastic, Plastic, and Creep Deformations of an Initially Isotropic Material Showing Anisotropic Strain-Hardening, Creep Recovery, and Secondary Creep." *ASME Journal of Applied Mechanics*. 25 (1958): 529-536.
4. Owen, D. R. J., A. Prakash, O. C. Zienkiewicz. "Finite Element Analysis of Non-Linear Composite Materials by Use of Overlay Systems." *Computers and Structures*. 4.6 (1974): 1251-1267.
5. Rice, J. R. "Continuum mechanics and thermodynamics of plasticity in relation to microscale deformation mechanisms." *Constitutive Equations in Plasticity*. Ed. A. Argon. Cambridge, MA: MIT Press, 1975. 23-79.
6. Chaboche, J. L. "Constitutive Equations for Cyclic Plasticity and Cyclic Viscoplasticity." *International Journal of Plasticity*. 5.3 (1989): 247-302.
7. Chaboche, J. L. "On Some Modifications of Kinematic Hardening to Improve the Description of Ratchetting Effects." *International Journal of Plasticity*. 7.7 (1991): 661-678.
8. Shih, C. F., D. Lee. "Further Developments in Anisotropic Plasticity." *Journal of Engineering Materials and Technology*. 100.3 (1978): 294-302.

9. Valliappan, S., P. Boonlaulohr, I. K. Lee. "Non-Linear Analysis for Anisotropic Materials." *International Journal for Numerical Methods in Engineering*. 10.3 (1976): 597-606.
10. Drucker, D. C., W. Prager. "Soil Mechanics and Plastic Analysis or Limit Design." *Quarterly of Applied Mathematics*. 10.2 (1952): 157-165.
11. Sandler, I. S., F. L. DiMaggio, G. Y. Baladi. "A Generalized Cap Model for Geological Materials." *Journal of the Geotechnical Engineering Division*. 102.7 (1976): 683-699.
12. Schwer, L. E., Y. D. Murray. "A Three-Invariant Smooth Cap Model with Mixed Hardening." *International Journal for Numerical and Analytical Methods in Geomechanics*. 18.10 (1994): 657-688.
13. Foster, C., R. Regueiro, A. Fossum, R. Borja. "Implicit Numerical Integration of a Three-Invariant, Isotropic/Kinematic Hardening Cap Plasticity Model for Geomaterials." *Computer Methods in Applied Mechanics and Engineering*. 194.50-52 (2005): 5109-5138.
14. Pelessone, D. "A Modified Formulation of the Cap Model." Technical Report GA-C19579. San Diego: Gulf Atomics, 1989.
15. Fossum, A. F., J. T. Fredrich. "Cap Plasticity Models and Compactive and Dilatant Pre-Failure Deformation." *Pacific Rocks 2000: Rock Around the Rim*. Proceedings of the Fourth North American Rock Mechanics Symposium. Eds. J. Girard, M. Liebman, C. Breeds, T. Doe, A. A. Balkema. Rotterdam, 2000: 1169-1176.
16. Gurson, A. L. "Continuum Theory of Ductile Rupture by Void Nucleation and Growth: Part I--Yield Criteria and Flow Rules for Porous Ductile Media." *Journal of Engineering Materials and Technology*. 99.1 (1977): 2-15.
17. Needleman, A. V. Tvergaard. "An Analysis of Ductile Rupture in Notched Bars." *Journal of the Mechanics and Physics of Solids*. 32.6 (1984): 461-490.
18. Arndt, S., B. Svendsen, D. Klingbeil. "Modellierung der Eigenspannungen an der Rißspitze mit einem Schädigungsmodell." *Technische Mechanik*. 4.17 (1997): 323-332.
19. Besson, J., C. Guillemer-Neel. "An Extension of the Green and Gurson Models to Kinematic Hardening." *Mechanics of Materials*. 35.1-2 (2003): 1-18.
20. Hjelm, H. E. "Yield Surface for Grey Cast Iron Under Biaxial Stress." *Journal of Engineering Materials and Technology*. 116.2 (1994): 148-154.
21. Chen, W. F., D. J. Han. *Plasticity for Structural Engineers*. New York: Springer-Verlag, 1988.
22. M hlich, U. and W. Brocks. "On the Numerical Integration of a Class of Pressure-Dependent Plasticity Models Including Kinematic Hardening." *Computational Mechanics*. 31.6 (2003): 479-488.

4.4.2. Isotropic Hardening

During plastic deformation, isotropic hardening causes a uniform increase in the size of the yield surface and results in an increase in the yield stress. The yield criterion has the form:

$$F(\sigma) - \sigma_y(\xi) = 0$$

where $F(\sigma)$ is a scalar function of stress and $\sigma_y(\xi)$ is the yield stress that evolves as a function of the set of material internal variables ξ . This type of hardening can model the behavior of materials

under monotonic loading and elastic unloading, but often does not give good results for structures that experience additional plastic deformation after a load reversal from a plastic state.

Three general [classes of isotropic hardening models](#) (p. 40) are available: bilinear, multilinear, and nonlinear. Each of the hardening models assumes a von Mises [yield criterion](#) (p. 37), unless an anisotropic Hill yield criterion is defined, and includes an associated flow rule.

Isotropic hardening can also be combined with [kinematic hardening](#) (p. 46) and the [Extended Drucker-Prager](#) (p. 55) and [Gurson](#) (p. 64) models to provide an evolution of the yield stress. For more information, see [Material Model Combinations](#) (p. 297).

The following topics concerning the isotropic hardening material model are available:

[4.4.2.1. Yield Criteria and Plastic Potentials](#)

[4.4.2.2. General Isotropic Hardening Classes](#)

4.4.2.1. Yield Criteria and Plastic Potentials

Hardening models assume a von Mises yield criterion, unless an anisotropic Hill yield criterion is defined.

[4.4.2.1.1. Von Mises Yield Criterion](#)

[4.4.2.1.2. Hill Yield Criterion](#)

4.4.2.1.1. Von Mises Yield Criterion

The von Mises yield criterion is commonly used in plasticity models for a wide range of materials. It is a good first approximation for metals, polymers, and saturated geological materials. The criterion is isotropic and independent of hydrostatic pressure, which can limit its applicability to microstructured materials and materials that exhibit plastic dilatation.

The von Mises yield criterion is:

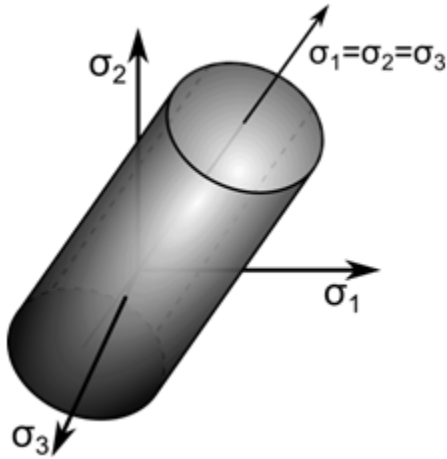
$$f(\sigma, \sigma_y) = \sigma_e - \sigma_y = 0 \quad (4.5)$$

where σ_e is the von Mises effective stress, also known as the von Mises equivalent stress,

$$\sigma_e = \sqrt{\frac{3}{2} \left(\sigma : \sigma - \frac{1}{3} \text{tr}(\sigma)^2 \right)}$$

and σ_y is the yield strength and corresponds to the yield in uniaxial stress loading.

In principal stress space, the yield surface is a cylinder with the axis along the hydrostatic line $\sigma_1 = \sigma_2 = \sigma_3$ and gives a yield criterion that is independent of the hydrostatic stress, as shown in the following figure:

Figure 4.6: Yield Surface for von Mises Yield Criterion

For an associated flow rule, the plastic potential is the yield criterion in [Equation 4.5 \(p. 37\)](#) and the plastic strain increment is proportional to the deviatoric stress

$$d\varepsilon^{pl} = d\lambda \left(\sigma - \frac{1}{3} \text{tr}(\sigma) \right)$$

4.4.2.1.2. Hill Yield Criterion

The Hill yield criterion [1] is an anisotropic criterion that depends on the orientation of the stress relative to the axis of anisotropy. It can be used to model materials in which the microstructure influences the macroscopic behavior of the material such as forged metals and composites.

In a coordinate system that is aligned with the anisotropy coordinate system, the Hill yield criterion given in stress components is:

$$\begin{aligned} f(\sigma, \sigma_y) = & F(\sigma_{22} - \sigma_{33})^2 + G(\sigma_{33} - \sigma_{11})^2 + H(\sigma_{11} - \sigma_{22})^2 \\ & + 2L\sigma_{23}^2 + 2M\sigma_{31}^2 + 2N\sigma_{12}^2 - \sigma_y^2 = 0 \end{aligned} \quad (4.6)$$

The coefficients in this yield criterion are functions of the ratio of the scalar yield stress parameter and the yield stress in each of the six stress components:

$$F = \frac{1}{2} \left(\frac{1}{R_{22}^2} + \frac{1}{R_{33}^2} - \frac{1}{R_{11}^2} \right)$$

$$G = \frac{1}{2} \left(\frac{1}{R_{33}^2} + \frac{1}{R_{11}^2} - \frac{1}{R_{22}^2} \right)$$

$$H = \frac{1}{2} \left(\frac{1}{R_{11}^2} + \frac{1}{R_{22}^2} - \frac{1}{R_{33}^2} \right)$$

$$L = \frac{3}{2} \left(\frac{1}{R_{23}^2} \right)$$

$$M = \frac{3}{2} \left(\frac{1}{R_{13}^2} \right)$$

$$N = \frac{3}{2} \left(\frac{1}{R_{12}^2} \right)$$

where the directional yield stress ratios are the user-input parameters and are related to the isotropic yield stress parameter by:

$$\begin{aligned} R_{11} &= \frac{\sigma_{11}^y}{\sigma_y} \\ R_{22} &= \frac{\sigma_{22}^y}{\sigma_y} \\ R_{33} &= \frac{\sigma_{33}^y}{\sigma_y} \\ R_{12} &= \sqrt{3} \frac{\sigma_{12}^y}{\sigma_y} \\ R_{23} &= \sqrt{3} \frac{\sigma_{23}^y}{\sigma_y} \\ R_{13} &= \sqrt{3} \frac{\sigma_{13}^y}{\sigma_y} \end{aligned}$$

where σ_i^y is the yield stress in the direction indicated by the value of subscript i. The stress directions are in the anisotropy coordinate system which is aligned with the element coordinate system (**ESYS**). The isotropic yield stress σ_y is entered in the constants for the hardening model.

The Hill yield criterion defines a surface in six-dimensional stress space and the flow direction is normal to the surface. The plastic strain increments in the anisotropy coordinate system are:

$$\begin{aligned} d\varepsilon_{11}^{pl} &= d\lambda [H(\sigma_1 - \sigma_2) + G(\sigma_1 - \sigma_3)] \\ d\varepsilon_{22}^{pl} &= d\lambda [F(\sigma_2 - \sigma_3) + H(\sigma_2 - \sigma_1)] \\ d\varepsilon_{33}^{pl} &= d\lambda [G(\sigma_3 - \sigma_1) + F(\sigma_3 - \sigma_2)] \\ d\varepsilon_{23}^{pl} &= d\lambda [2L\sigma_{23}] \\ d\varepsilon_{31}^{pl} &= d\lambda [2M\sigma_{31}] \\ d\varepsilon_{12}^{pl} &= d\lambda [2N\sigma_{12}] \end{aligned}$$

The Hill surface, used with a hardening model, replaces the default von Mises yield surface.

After defining the material data table (**TB,HILL**), input the required constants (**TBDATA**):

Constant	Meaning	Property
C1	R ₁₁	Yield stress ratio in X direction
C2	R ₂₂	Yield stress ratio in Y direction
C3	R ₃₃	Yield stress ratio in Z direction
C4	R ₁₂	Yield stress ratio in XY direction
C5	R ₂₃	Yield stress ratio in YZ direction
C6	R ₁₃	Yield stress ratio in XZ direction

The constants can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

Example 4.2: Hill Surface

```
/prep7
MP, EX, 1, 20.0E5
MP, NUXY, 1, 0.3
! ELASTIC CONSTANTS
```

```
TB,HILL,1
TBDATA,1,1.0,1.1,0.9,0.85,0.9,0.80 ! HILL TABLE
```

4.4.2.1.2.1. Separated Hill Potentials for Plasticity and Creep

This capability is reserved for use with the material combination of Chaboche nonlinear kinematic hardening with implicit creep.

To define separated Hill potentials for the plastic yielding and the creep flow (**TB,HILL,,,,,PC**), issue two **TBDATA** commands, one to set constants C1-C6 defining the Hill yield surface for plasticity, the other to set constants C7-C12 defining the Hill potential for the creep direction.

Example 4.3: Defining Hill Surfaces for Plasticity and Creep

```
/prep7
MP,EX,1,20.0E5 ! ELASTIC CONSTANTS
MP,NUXY,1,0.3

TB,HILL,1,1,,PC ! HILL data table for plasticity and creep
TBDATA,1,1.0,1.1,0.9,0.85,0.9,0.80 ! Plasticity Hill parameters
TBDATA,7,1.0,1.0,1.0,1.0,1.0,1.0 ! Creep Hill parameters
```

4.4.2.2. General Isotropic Hardening Classes

Support is available for these general classes of isotropic hardening:

4.4.2.2.1. Bilinear Isotropic Hardening

4.4.2.2.2. Multilinear Isotropic Hardening

4.4.2.2.3. Nonlinear Isotropic Hardening

4.4.2.2.4. Isotropic Hardening Static Recovery

4.4.2.2.1. Bilinear Isotropic Hardening

Bilinear isotropic hardening is described by a bilinear effective-stress versus effective-strain curve. The initial slope of the curve is the elastic modulus of the material. Beyond the user-specified initial yield stress σ_0 , plastic strain develops and stress-vs.-total-strain continues along a line with slope defined by the user-specified tangent modulus E_T . The tangent modulus cannot be less than zero or greater than the elastic modulus.

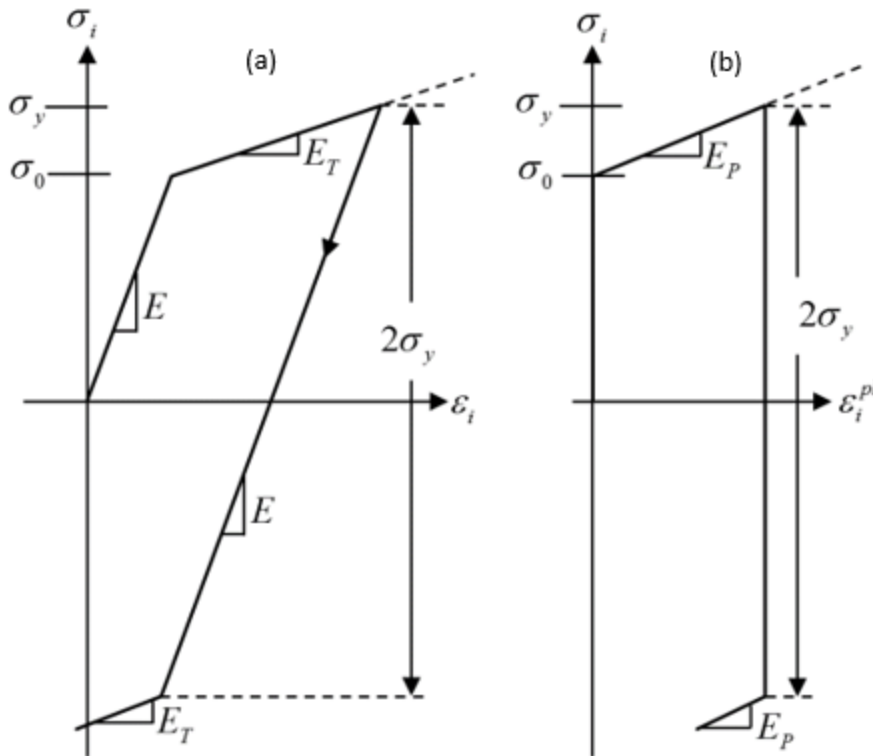
In effective-stress vs. effective-plastic-strain space, bilinear isotropic hardening is described by a line initially intersecting the stress axis at σ_0 and continuing with the slope of the user-defined plastic tangent modulus E_p .

The tangent modulus E_T and the plastic tangent modulus E_p are related by:

$$E_p = \frac{EE_T}{E - E_T}$$

where E is the elastic modulus.

Figure 4.7: Stress vs. Total Strain (a), and Stress vs. Plastic Strain (b) for Bilinear Isotropic Hardening



4.4.2.2.1.1. Defining the Bilinear Isotropic Hardening Model

Define the isotropic or anisotropic elastic behavior via **MP** commands.

Bilinear isotropic hardening is defined by providing either the plastic tangent modulus E_p (**TB,PLAS,,,BISO**) or the tangent modulus E_T (**TB,BISO**).

The constants can be defined as a function of temperature (**TB,,,NTEMP**), with temperatures specified for the table entries (**TBTEMP**).

Defining the Bilinear Isotropic Hardening Model via **TB,PLAS,,,BISO**

After defining the material data table (**TB,PLAS,,,BISO**), input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	σ_0	Yield stress
C2	E_p	Plastic tangent modulus

Example 4.4: Bilinear Isotropic Hardening

```

/prep7
MPTEMP,1,0,500          ! Define temperatures for Young's modulus
MPDATA,EX,1,,14E6,12E6
MPDATA,PRXY,1,,0.3,0.3
TB,PLAS,1,2,,BISO        ! Activate a data table
TBTEMP,0.0               ! Temperature = 0.0
TBDATA,1,44E3,1.2E6       ! Yield = 44,000; Plastic Tangent modulus = 1.2E6
TBTEMP,500               ! Temperature = 500
TBDATA,1,29.33E3,0.8E6    ! Yield = 29,330; Plastic Tangent modulus = 0.8E6

```

Defining the Bilinear Isotropic Hardening Model via TB,BISO

After defining the material data table (TB,BISO), input the following constants (TB,DATA):

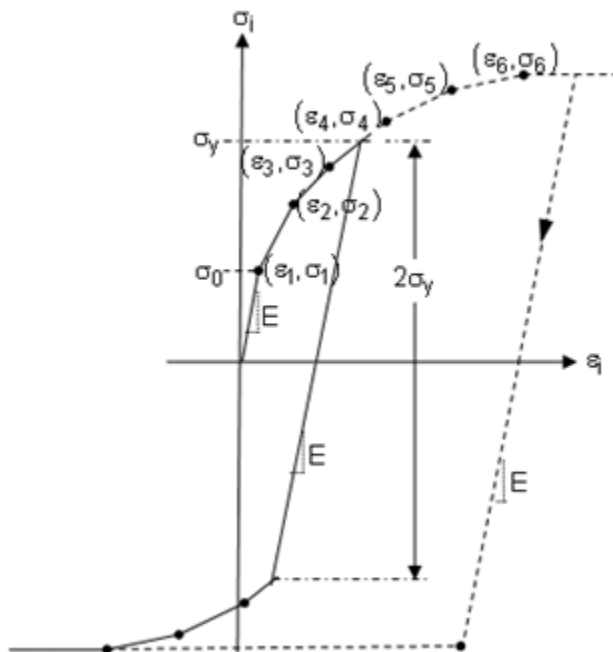
Constant	Meaning	Property
C1	σ_0	Yield stress
C2	E_T	Tangent modulus

4.4.2.2.2. Multilinear Isotropic Hardening

The behavior of multilinear isotropic hardening is similar to bilinear isotropic hardening except that a multilinear stress versus total or plastic strain curve is used instead of a bilinear curve.

The multilinear hardening behavior is described by a piece-wise linear stress-total strain curve, starting at the origin and defined by sets of positive stress and strain values, as shown in this figure:

Figure 4.8: Stress vs. Total Strain for Multilinear Isotropic Hardening



The first stress-strain point corresponds to the yield stress. Subsequent points define the elastic plastic response of the material.

4.4.2.2.2.1. Defining the Multilinear Isotropic Hardening Model

Define the isotropic or anisotropic elastic behavior via **MP** commands. After defining the material data table (TB,PLASTIC,,,MISO), input the following constants (TB,DATA):

Constant	Meaning	Property
X	ϵ_i^{pl}	Plastic strain value
Y	σ_i	Stress value

The stress-plastic strain data points are entered into the table via the **TBPT** command.

Temperature-dependent data can be defined (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**). Interpolation between temperatures occurs via stress-vs.-plastic-strain.

Example 4.5: Multilinear Hardening with Plastic Strain

```
/prep7
MPTEMP,1,0,500          ! Define temperature-dependent EX,
MPDATA,EX,1,,14.665E6,12.423E6
MPDATA,PRXY,1,,0.3

TB,PLASTIC,1,2,5,MISO    ! Activate TB,PLASTIC data table
TBTEMP,0.0              ! Temperature = 0.0
TBPT,DEFI,0,29.33E3      ! Plastic strain, stress at temperature = 0
TBPT,DEFI,1.59E-3,50E3
TBPT,DEFI,3.25E-3,55E3
TBPT,DEFI,5.91E-3,60E3
TBPT,DEFI,1.06E-2,65E3
TBTEMP,500              ! Temperature = 500
TBPT,DEFI,0,27.33E3      ! Plastic strain, stress at temperature = 500
TBPT,DEFI,2.02E-3,37E3
TBPT,DEFI,3.76E-3,40.3E3
TBPT,DEFI,6.48E-3,43.7E3
TBPT,DEFI,1.12E-2,47E3
```

4.4.2.2.3. Nonlinear Isotropic Hardening

Power law (p. 43) and Voce (p. 44) equations are available to model nonlinear isotropic hardening.

4.4.2.2.3.1. Power Law Nonlinear Isotropic Hardening

4.4.2.2.3.2. Voce Law Nonlinear Isotropic Hardening

4.4.2.2.3.1. Power Law Nonlinear Isotropic Hardening

The power law equation has a user-defined initial yield stress σ_0 and exponent N . The current yield stress σ_y is given by solving the following equation:

$$\frac{\sigma_y}{\sigma_0} = \left(\frac{\sigma_y}{\sigma_0} + \frac{3G}{\sigma_0} \varepsilon^{pl} \right)^N$$

where G is the shear modulus determined from the user-defined elastic constants and ε^{pl} is the accumulated equivalent plastic strain.

Defining the Power Law Nonlinear Isotropic Hardening Model

For the power law hardening model, define the isotropic or anisotropic elastic behavior via **MP** commands. After defining the material data table (**TB,NLISO,,,POWER**), input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	σ_0	Initial yield stress
C2	N	Exponent

The exponent N must be positive and less than 1.

Temperature-dependent data can be defined (*NTEMP* on the **TB** command), with temperatures specified for the subsequent set of constants (**TBTEMP**).

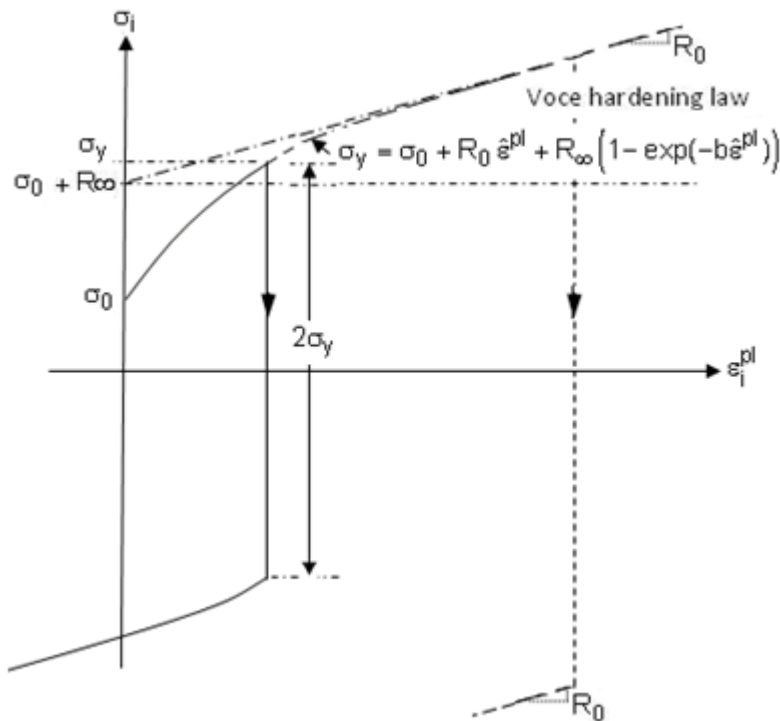
Example 4.6: Power Law Nonlinear Isotropic Hardening

```
/prep7
TB,NLISO,1,2,,POWER
TBTEMP,100           ! Define first temperature
TBDATA,1,275,0.1
TBTEMP,200           ! Define second temperature
TBDATA,1,275,0.1
```

4.4.2.2.3.2. Voce Law Nonlinear Isotropic Hardening

The Voce hardening model is similar to bilinear isotropic hardening, with an exponential saturation hardening term added to the linear term, as shown in this figure:

Figure 4.9: Stress vs. Plastic Strain for Voce Hardening



The evolution of the yield stress for this model is specified by the following equation:

$$\sigma_Y = \sigma_0 + R_0 \hat{\epsilon}^{pl} + R_\infty (1 - \exp(-b \hat{\epsilon}^{pl}))$$

where the user-defined parameters include R_∞ , the difference between the saturation stress and the initial yield stress, R_0 , the slope of the saturation stress and, b , the hardening parameter that governs the rate of saturation of the exponential term.

Defining the Voce Law Nonlinear Isotropic Hardening Model

Define the isotropic or anisotropic elastic behavior via **MP** commands. After defining the material data table (**TB,NLISO,,,VOCE**), input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	σ_0	Initial yield stress
C2	R_0	Linear coefficient
C3	R_∞	Exponential coefficient
C4	b	Exponential saturation parameter

The constants can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the subsequent set of constants (**TBTEMP**).

Example 4.7: Voce Nonlinear Isotropic Hardening

```
/PREP7
TB,NLISO,1,2,,VOCE      ! Activate NLISO data table
TBTEMP,40                ! Define first temperature
TBDATA,1,280,7e3,155,7e2 ! Constants at first temperature
TBTEMP,60                ! Define second temperature
TBDATA,1,250,5e3,120,3e2 ! Constants at second temperature
```

4.4.2.2.4. Isotropic Hardening Static Recovery

Static recovery, also known as thermal recovery, of the isotropic yield stress is dependent on time, temperature and the current yield stress. The rate of isotropic hardening can be separated into work hardening and static recovery, as follows:

$$\dot{\sigma}_y = R_w(\dot{\varepsilon}^{pl}, \dot{\varepsilon}^{pl}, \sigma_y) - r_1 \exp\left(-\frac{r_2}{T}\right) \sigma_y^{r_3}$$

where the first term on the right side of the equation represents the rate of work hardening, and the second defines the rate of static recovery where r_1 , r_2 and r_3 are material parameters, T is the temperature, and σ_y is the current yield stress. A lower limit yield stress can be defined for static recovery such that:

$$r_1 \exp\left(-\frac{r_2}{T}\right) \sigma_y^{r_3} = 0 \quad \text{for } \sigma_y < r_4$$

where r_4 is the user-defined static recovery threshold.

An approximate solution for the static recovery of the yield stress over a time step is to integrate the static recovery rate and incorporate the work hardening rate into the initial condition. The yield stress is then given by the solution to:

$$-\dot{\sigma}_y = r_1 \exp\left(-\frac{r_2}{T}\right) \sigma_y^{r_3}$$

with the initial condition:

$$\sigma_y(t_0) \equiv \sigma_y^0 + \Delta\sigma_y^w$$

where σ_y^0 is the yield stress at the end of the previous increment and $\Delta\sigma_y^w$ is the increment in yield stress due to work hardening over the current time step.

4.4.2.2.4.1. Defining the Isotropic Static Recovery

Static recovery of the isotropic yield stress can be used with the combined creep and Chaboche nonlinear hardening material. The material parameters are defined via a **TB** material table with *Lab* = PLASTIC and *TBOPT* = ISR.

Constant	Meaning	Property
C1	r_1	Coefficient
C2	r_2	Temperature Coefficient
C3	r_3	Exponential
C4	r_4	Recovery threshold

The constants can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

Example 4.8: Isotropic Hardening Static Recovery

```
/prep7
YOUNGS = 30e3    ! Young's Modulus
NU      = 0.3     ! Poisson's ratio
SIGMA0 = 18.0E0  ! Initial yield stress

mp,ex,1,YOUNGS
mp,prxy,1,NU

TB,CHAB,1,1,1    ! Chaboche kinematic hardening
TBDATA,1,sigma0,
TBDATA,2,1e+3,0

TB,CREEP,1,1, ,1,1 ! creep model
tbdata,1,         ! No creep strain

TB,PLASTIC,1,,MISO ! Multilinear isotropic hardening
TBPT,DEFI,0.0,SIGMA0
TBPT,DEFI,0.001,35.0
TBPT,DEFI,0.002,40.0
TBPT,DEFI,0.010,60.0

TB,PLASTIC,1,,4,ISR ! Isotropic hardening static recovery
TBDATA,1,1D-5,1,2,SIGMA0
```

4.4.3. Kinematic Hardening

During plastic deformation, kinematic hardening causes a shift in the yield surface in stress space. In uniaxial tension, plastic deformation causes the tensile yield stress to increase and the magnitude of the compressive yield stress to decrease. This type of hardening can model the behavior of materials under either monotonic or cyclic loading and can be used to model phenomena such as the Bauschinger effect and plastic ratcheting.

The yield criterion has the form:

$$F(\bar{\sigma}) - \sigma_y = 0$$

where $F(\bar{\sigma})$ is a scalar function of the relative stress $\bar{\sigma}$ and σ_y is the yield stress. The relative stress is:

$$\bar{\sigma} = \sigma - \alpha \quad (4.7)$$

where the backstress α is the shift in the position of the yield surface in stress space and evolves during plastic deformation.

The general classes of [kinematic hardening models](#) (p. 47) are bilinear, multilinear, and nonlinear. The hardening models assume a von Mises [yield criterion](#) (p. 37), unless an anisotropic Hill yield criterion is defined, and includes an associated flow rule.

Kinematic hardening can also be combined with [isotropic hardening](#) (p. 36) and the [Gurson](#) (p. 64) model to provide an evolution of the yield stress. For more information, see [Material Model Combinations](#) (p. 297).

The following topics concerning the kinematic hardening material model are available:

[4.4.3.1. Yield Criteria and Plastic Potentials](#)

[4.4.3.2. General Kinematic Hardening Classes](#)

[4.4.3.3. Kinematic Hardening Static Recovery](#)

4.4.3.1. Yield Criteria and Plastic Potentials

Kinematic hardening uses the von Mises yield criterion with an associated flow rule unless a Hill yield surface is defined. If a Hill yield surface is defined, the kinematic hardening model uses it for both the yield criterion and as the plastic potential to give an associated flow rule.

For more information about von Mises and Hill yield surfaces, see [Yield Criteria and Plastic Potentials](#) (p. 37).

4.4.3.2. General Kinematic Hardening Classes

Support is available for these general classes of kinematic hardening:

[4.4.3.2.1. Bilinear Kinematic Hardening](#)

[4.4.3.2.2. Multilinear Kinematic Hardening](#)

[4.4.3.2.3. Nonlinear Kinematic Hardening](#)

4.4.3.2.1. Bilinear Kinematic Hardening

The backstress tensor for bilinear kinematic hardening evolves so that the effective stress versus effective strain curve is bilinear. The initial slope of the curve is the elastic modulus of the material and beyond the user-specified initial yield stress σ_0 , plastic strain develops and the backstress evolves so that stress versus total strain continues along a line with slope defined by the user-specified tangent modulus E_T . This tangent modulus cannot be less than zero or greater than the elastic modulus.

In effective-stress vs. effective-plastic-strain space, bilinear kinematic hardening is described by a line initially intersecting the stress axis at σ_0 and continuing with the slope of the user-defined plastic tangent modulus E_p .

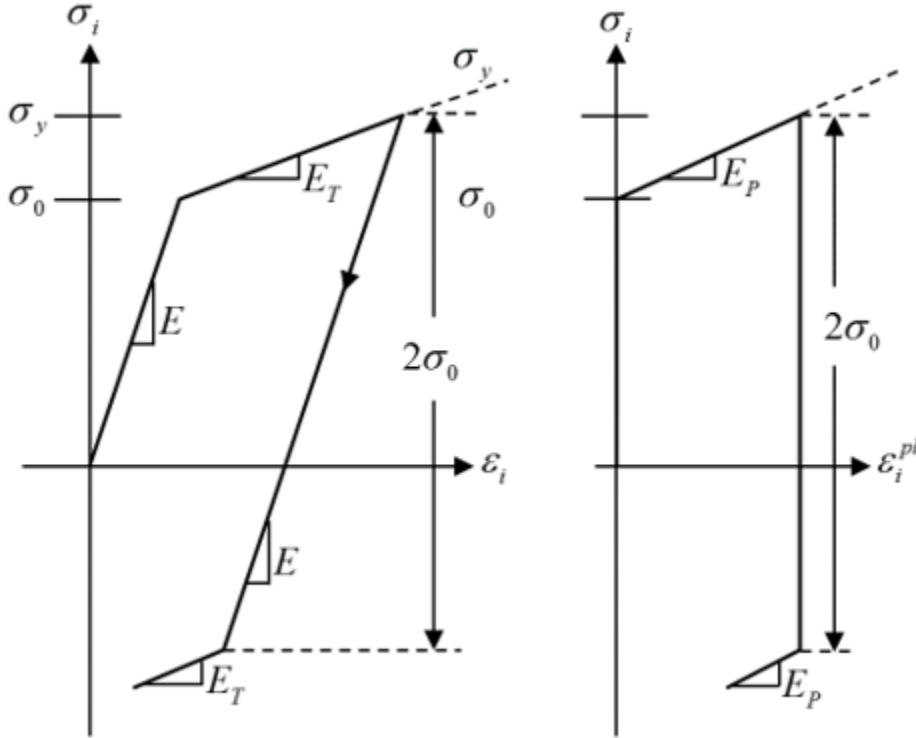
The tangent modulus E_T and the plastic tangent modulus E_p are related by:

$$E_p = \frac{EE_T}{E - E_T}$$

where E is the elastic modulus.

For uniaxial tension followed by uniaxial compression, the magnitude of the compressive yield stress decreases as the tensile yield stress increases so that the magnitude of the elastic range is always $2\sigma_0$, as shown in this figure:

Figure 4.10: Stress vs. Total Strain (a), and Stress vs. Plastic Strain (b) for Bilinear Kinematic Hardening



The backstress is proportional to the shift strain ε^{sh} :

$$\alpha = 2G\varepsilon^{sh}$$

where G is the elastic shear modulus and the shift strain is numerically integrated from the incremental shift strain which is proportional to the incremental plastic strain:

$$d\varepsilon^{sh} = \frac{C}{2G} d\varepsilon^{pl}$$

where

$$C = \frac{2}{3} \frac{EE_T}{E - E_T} = \frac{2}{3} E_P$$

and E is Young's Modulus, E_T is the user-defined tangent modulus [2], and E_P is the user-defined plastic tangent modulus. The incremental plastic strain is defined by the associated flow rule for the von Mises or Hill potential given in [Yield Criteria and Plastic Potentials \(p. 37\)](#) with the stress given by the relative stress $\bar{\sigma}$.

4.4.3.2.1.1. Defining the Bilinear Kinematic Hardening Model

Define the isotropic or anisotropic elastic behavior via **MP** commands.

Bilinear kinematic hardening is defined by providing either the plastic tangent modulus E_p (**TB,PLAS,,,,BKIN**) or the tangent modulus E_T (**TB,BKIN**).

The constants can be defined as a function of temperature (**TB,,,NTEMP**), with temperatures specified for the table entries (**TBTEMP**).

Defining the Bilinear Kinematic Hardening Model via **TB,PLAS,,,,BKIN**

After defining the material data table (**TB,PLAS,,,,BKIN**), input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	σ_0	Initial yield stress
C2	E_p	Plastic tangent modulus

Example 4.9: Bilinear Kinematic Hardening

```
/prep7
MPTEMP,1,0,500          ! Define temperatures for Young's modulus
MPDATA,EX,1,,14E6,12e6
MPDATA,PRXY,1,,0.3,0.3
TB,PLAS,1,2,2,BKIN      ! Activate a data table
TBTEMP,0.0              ! Temperature = 0.0
TBDATA,1,44E3,1.2E6     ! Yield = 44,000; Plastic Tangent modulus = 1.2E6
TBTEMP,500              ! Temperature = 500
TBDATA,1,29.33E3,0.8E6  ! Yield = 29,330; Plastic Tangent modulus = 0.8E6
```

Defining the Bilinear Kinematic Hardening Model via **TB,BKIN**

After defining the material data table (**TB,BKIN**), input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	σ_0	Initial yield stress
C2	E_T	Tangent modulus

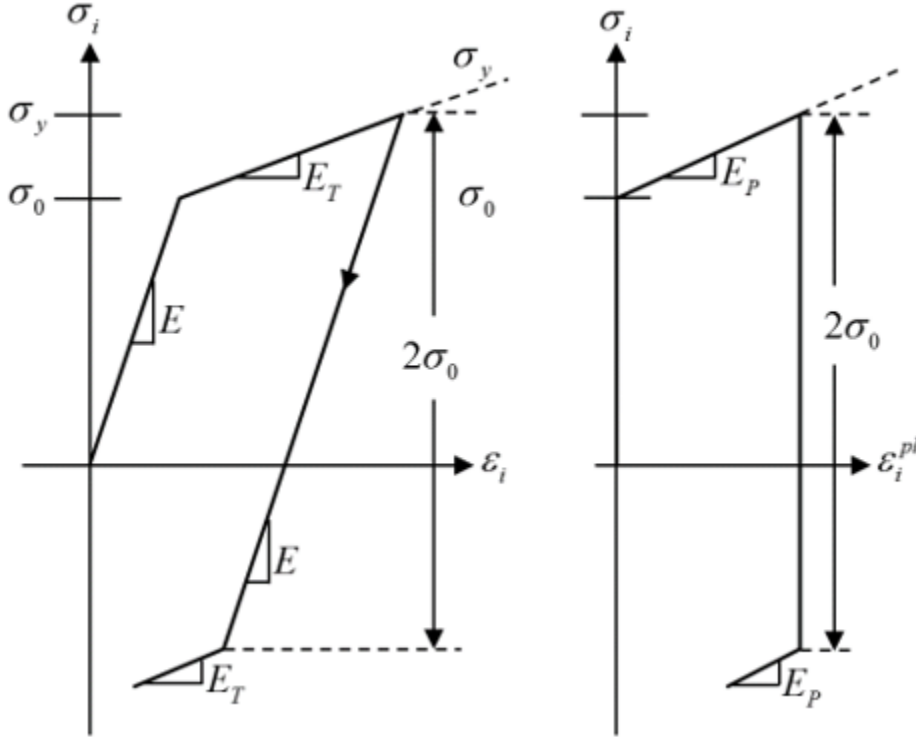
This model can be used with **TB,,,,TBOPT**:

For $TBOPT \neq 1$, no stress relaxation occurs with an increase in temperature. This option is not recommended for non-isothermal problems.

For $TBOPT = 1$, [Rice's hardening rule \(p. 35\)](#) is applied, which accounts for stress relaxation with temperature increase.

4.4.3.2.2. Multilinear Kinematic Hardening

The backstress tensor for multilinear kinematic hardening evolves so that the effective stress versus effective strain curve is multilinear with each of the linear segments defined by a set of user input stress-strain points, as shown in this figure:

Figure 4.11: Stress vs. Total Strain for Multilinear Kinematic Hardening

The model formulation is the sublayer or overlay model of [Besselling \(p. 35\)](#) and [Owen, Prakash and Zienkiewicz \(p. 35\)](#) in which the material is assumed to be composed of a number of sublayers or subvolumes, all subjected to the same total strain. The number of subvolumes is the same as the number of input stress-strain points, and the overall behavior is weighted for each subvolume where the weight is given by:

where E_{Tk} is the tangent modulus for segment of the stress-strain curve.

$$w_k = \frac{E - E_{Tk}}{E - \frac{1-2\nu}{3} E_{Tk}} - \sum_{i=1}^{k-1} w_i$$

The behavior of each subvolume is elastic-perfectly plastic, with the uniaxial yield stress for each subvolume given by:

$$\sigma_{yk} = \frac{1}{2(1+\nu)} (3E\epsilon_k - (1-2\nu)\sigma_k)$$

where (ϵ_k, σ_k) is the input stress-strain point for subvolume k .

For plane stress, $\nu=0.5$ is used to calculate the weights and subvolume yield stresses. The resulting stress-strain behavior is exact for uniaxial stress states but can deviate from the defined stress-strain values for general deformations.

The default yield surface is the von Mises surface, and each subvolume yields at an equivalent stress equal to the subvolume uniaxial yield stress. A Hill yield criterion can be used in which each subvolume yields according to the Hill criterion with the subvolume uniaxial yield as the isotropic yield stress and the subvolume anisotropic yield condition determined by the Hill surface.

The subvolumes undergo kinematic hardening with an associated flow rule and the plastic strain increment for each subvolume is the same as that for bilinear kinematic hardening (**TB,BKIN**). The total plastic strain is given by:

$$d\epsilon^{pl} = \sum_{i=1}^{N_{sv}} w_i d\epsilon_i^{pl}$$

For more information, see [Specialization for Multilinear Kinematic Hardening in the Theory Reference](#).

4.4.3.2.2.1. Defining the Multilinear Kinematic Hardening Model

Define the isotropic or anisotropic elastic behavior via **MP** commands. To specify the hardening behavior, define the material data table (**TB,PLAS,,,,KINH**) and input the constants (**TBPT**) as stress vs. total strain points or as stress vs. plastic strain points.

4.4.3.2.2.2. Specifying the Constants

Constant	Meaning	Property
P1	ϵ_i	Strain value
P2	σ_i	Stress value

The constants can be defined as a function of temperature (**NTEMP** on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

When entering temperature-dependent stress-strain points, the set of data at each temperature must have the same number of points. Thermal softening for the multilinear kinematic hardening model is the same as that for bilinear kinematic hardening (**TB,BKIN**) with [Rice's hardening rule](#) (p. 35).

Entering Stress vs. Plastic Strain Points

After defining the material data table (**TB,PLASTIC,,,,KINH**), enter the stress-strain points (**TBPT**).

No segment slope can be larger than the slope of the previous segment.

Example 4.10: Multilinear Kinematic Hardening with Stress vs. Plastic Strain

```

/prep7
TB,PLASTIC,1,2,3,KINH      ! Define the material data table
TBTEMP,20.0                ! Temperature = 20.0
TBPT,,0.0,1.0              ! Plastic Strain = 0.0000, Stress = 1.0
TBPT,,0.1,1.2              ! Plastic Strain = 0.1000, Stress = 1.2
TBPT,,0.2,1.3              ! Plastic Strain = 0.2000, Stress = 1.3
TBTEMP,40.0                ! Temperature = 40.0
TBPT,,0.0,0.9              ! Plastic Strain = 0.0000, Stress = 0.9
TBPT,,0.0900,1.0           ! Plastic Strain = 0.0900, Stress = 1.0
TBPT,,0.129,1.05           ! Plastic Strain = 0.1290, Stress = 1.05

```

4.4.3.2.3. Nonlinear Kinematic Hardening

The nonlinear kinematic hardening model is a rate-independent version of the kinematic hardening model proposed by Chaboche [6][7]. The model allows the superposition of several independent back-stress tensors and can be combined with any of the available isotropic

hardening models. It can be useful in modeling cyclic plastic behavior such as cyclic hardening or softening and ratcheting or shakedown.

The model uses an associated flow rule with either the default [von Mises \(p. 37\)](#) yield criterion or the [Hill \(p. 38\)](#) yield criterion if it is defined. The relative stress given by [Equation 4.4 \(p. 31\)](#) is used to evaluate the yield function, and the back-stress tensor is given by the superposition of a number of evolving kinematic back-stress tensors:

$$\alpha = \sum_{i=1}^n \alpha_i \quad (4.8)$$

where n is the number of kinematic models to be superposed.

The evolution of each back-stress model in the superposition is given by the kinematic hardening rule:

$$\dot{\alpha}_i = \frac{2}{3} C_i \dot{\epsilon}^{pl} - \gamma_i \dot{\epsilon}^{pl} \alpha_i$$

where C_i and γ_i are user-input material parameters, $\dot{\epsilon}^{pl}$ is the plastic strain rate, and $\dot{\epsilon}^{pl}$ is the magnitude of the plastic strain rate.

During a solution, if there is a change in temperature over an increment (non-isothermal loading), the back-stress terms are scaled in a manner similar to that of [bilinear kinematic hardening \(p. 47\)](#) with Rice's hardening rule [5]. The scaling redefines the back-stress at the beginning of the increment as:

$$\alpha_i(t_0) \rightarrow \frac{C_i(\theta)\gamma_i(\theta_0)}{C_i(\theta_0)\gamma_i(\theta)} \alpha_i(t_0)$$

where t_0 is the time at the beginning of the increment, θ_0 is the temperature at the beginning of the increment, and θ is the temperature at the end of the increment. Then the incremental back-stress update uses the material parameters at the current temperature, but is otherwise unchanged from the isothermal update.

For use with kinematic static recovery (see below), the non-isothermal back-stress evolution is given by:

$$\dot{\alpha}_i = \frac{2}{3} C_i \dot{\epsilon}^{pl} - \gamma_i \dot{\epsilon}^{pl} \alpha_i + \frac{1}{C_i} \frac{dC_i}{d\theta} \dot{\theta} \alpha_i \quad (4.9)$$

4.4.3.2.3.1. Defining the Nonlinear Kinematic Hardening Model

Define the material data table (**TB,CHABOCHE**) and input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	σ_0	Initial yield stress
C2	C_1	Material constant for first kinematic model
C3	γ_1	Material constant for first kinematic model

Constant	Meaning	Property
C4	C_2	Material constant for second kinematic model
C5	γ_2	Material constant for second kinematic model
...	...	...
C(2n)	C_n	Material constant for last kinematic model
C(1+2n)	γ_n	Material constant for last kinematic model

Define temperature-dependent data via *NTEMP*, with temperatures specified for the table entries (**TBTEMP**).

Set *NPTS* = *n*, the number of superimposed kinematic hardening models.

To use Equation 4.9 (p. 52) with kinematic static recovery, set *TBOPT* = TRATE. With this option, the material parameters are the same as those given in the table above.

Example 4.11: Nonlinear Kinematic Hardening

```
/prep7
TB,CHABOCHE,1,1,3      ! Activate Chaboche data table with
                        ! 3 models to be superposed
! Define Chaboche material data
TBDATA,1,18.8           ! C1 - Initial yield stress
TBDATA,2,5174000,4607500 ! C2,C3 - Chaboche constants for 1st model
TBDATA,4,17155,1040     ! C4,C5 - Chaboche constants for 2nd model
TBDATA,6,895.18,9       ! C6,C7 - Chaboche constants for 3rd model
```

4.4.3.3. Kinematic Hardening Static Recovery

Static recovery (also known as thermal recovery) for kinematic hardening is included in the Chaboche nonlinear kinematic hardening model by modifying the evolution of the back-stress tensor components:

$$\dot{\alpha}_i = \frac{2}{3} C_i \dot{\epsilon}^{pl} - \gamma_i \dot{\epsilon}^{pl} \alpha_i + \frac{1}{C_i} \frac{dC_i}{d\theta} \dot{\theta} \alpha_i - \frac{C_i}{M_i^{m_i}} \bar{\alpha}_i^{m_i-1} \alpha_i$$

where the last term on the right side of the equation is the rate of static recovery of the kinematic back-stress component with M_i and m_i the material parameters for kinematic static recovery, and $\bar{\alpha}_i$ is the von Mises effective back-stress component.

4.4.3.3.1. Defining the Kinematic Static Recovery

Static recovery of the kinematic backstress can be used with either of the following material property combinations:

- Creep and Chaboche nonlinear hardening
- Viscoplastic and Chaboche nonlinear hardening

The temperature-rate form of the back-stress evolution is required (**TB,CHAB,,,,TRATE**).

Use a material data table to define the material parameters (**TB,PLASTIC,,,,KSR2**). Ensure that the number of material parameter sets corresponds to the number of superimposed Chaboche back-stress terms. Undefined values default to 0.0.

Constant	Meaning	Property
C1	M_i	Coefficient
C2	m_i	Exponent

The constants can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

Example 4.12: Kinematic Hardening Static Recovery

```
/prep7

YOUNGS = 30e3    ! Young's Modulus
NU      = 0.3     ! Poisson's ratio
SIGMA0 = 18.0E0 ! Initial yield stress

mp,ex,1,YOUNGS
mp,prxy,1,NU

TB,CHAB,1,1,1,TRATE ! Chaboche kinematic hardening
TB,DATA,1,sigma0,
TB,DATA,2,1e+4,0

TB,PLASTIC,1,,1,KSR2 ! Kinematic hardening static recovery
TB,DATA,1,1e3,2.0,
```

4.4.4. Drucker-Prager

The following topics concerning to Drucker-Prager plasticity are available:

4.4.4.1. Classic Drucker-Prager

4.4.4.2. Extended Drucker-Prager (EDP)

4.4.4.3. Extended Drucker-Prager Cap

4.4.4.1. Classic Drucker-Prager

The classic Drucker-Prager model [10] is applicable to granular (frictional) material such as soils, rock, and concrete and uses the outer cone approximation to the [Mohr-Coulomb law](#). The input consists of only three constants:

- Cohesion value (> 0)
- Angle of internal friction
- Dilatancy angle

The amount of dilatancy (the increase in material volume due to yielding) can be controlled via the dilatancy angle. If the dilatancy angle is equal to the friction angle, the flow rule is associative. If the dilatancy angle is zero (or less than the friction angle), there is no (or less of an) increase in material volume when yielding and the flow rule is non-associated.

For more information about this material model, see [Classic Drucker-Prager Model](#) in the [Mechanical APDL Theory Reference](#).

4.4.4.1.1. Defining the Classic Drucker-Prager Model

Define the isotropic or anisotropic elastic behavior via **MP** commands. Define the material data table (**TB,DP**) and define up to three constants (**TB,DATA**), as follows:

Constant	Meaning	Property
C1	Force/Area	Cohesion value
C2	Angle (in degrees)	Internal friction
C3	Angle (in degrees)	Dilatancy

Temperature-dependent parameters are not allowed.

Example 4.13: Classic Drucker-Prager

```
MP,EX,1,5000
MP,NUXY,1,0.27
TB,DP,1
TB,DATA,1,2.9,32,0    ! Cohesion = 2.9 (use consistent units),
                      ! Angle of internal friction = 32 degrees,
                      ! Dilatancy angle = 0 degrees
```

4.4.4.2. Extended Drucker-Prager (EDP)

The extended Drucker-Prager (EDP) material model includes three yield criteria and corresponding flow potentials similar to those of the [classic Drucker-Prager \(p. 54\)](#) model commonly used for geomaterials with internal cohesion and friction. The yield functions can also be combined with an isotropic or kinematic hardening rule to evolve the yield stress during plastic deformation.

The model is defined via one of the three [yield criteria \(p. 55\)](#) combined with *any* of the three [flow potentials \(p. 58\)](#) and an optional hardening model.

The following topics for defining the EDP material model are available:

4.4.4.2.1. EDP Yield Criteria Forms

4.4.4.2.2. EDP Plastic Flow Potentials

4.4.4.2.3. Plastic Strain Increments for Flow Potentials

4.4.4.2.4. Example EDP Material Model Definitions

4.4.4.2.1. EDP Yield Criteria Forms

The EDP yield criteria include the following forms:

4.4.4.2.1.1. Linear Form

4.4.4.2.1.2. Power Law Form

4.4.4.2.1.3. Hyperbolic Form

4.4.4.2.1.1. Linear Form

The EDP linear yield criterion form is:

$$f(\sigma, \sigma_y) = \sigma_e + \alpha \frac{1}{3} \text{tr}(\sigma) - \sigma_y = 0$$

where the user-defined parameters are the pressure sensitivity α and the uniaxial yield stress σ_y .

Defining the EDP Linear Yield Criterion

After initializing the extended Drucker-Prager linear yield criterion (**TB**,EDP,,,,LYFUN), enter the following constants (**TB**DATA):

Constant	Meaning	Property
C1	α	Pressure sensitivity
C2	σ_y	Uniaxial yield stress

The constants can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TB**TEMP).

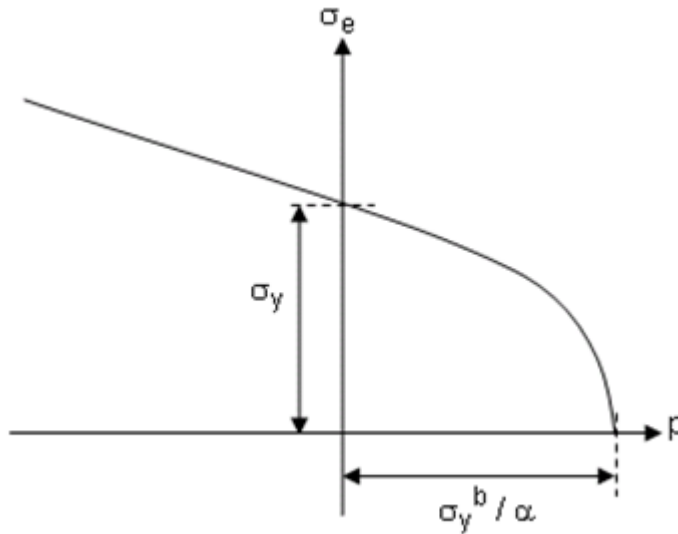
4.4.4.2.1.2. Power Law Form

The EDP power law yield criteria form is:

$$f(\sigma, \sigma_y) = \sigma_e^b + \alpha \frac{1}{3} \text{tr}(\sigma) - \sigma_y^b = 0$$

where the exponent b , pressure sensitivity α , and uniaxial yield stress σ_y are the user-defined parameters.

Figure 4.12: Power Law Criterion in the Meridian Plane



Defining the EDP Power Law Yield Criterion

After initializing the extended Drucker-Prager power law yield criterion (**TB**,EDP,,,,PYFUN), enter the following constants (**TB**DATA):

Constant	Meaning	Property
C1	α	Pressure sensitivity
C2	b	Exponent

Constant	Meaning	Property
C3	σ_y	Uniaxial yield stress

The constants can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

4.4.4.2.1.3. Hyperbolic Form

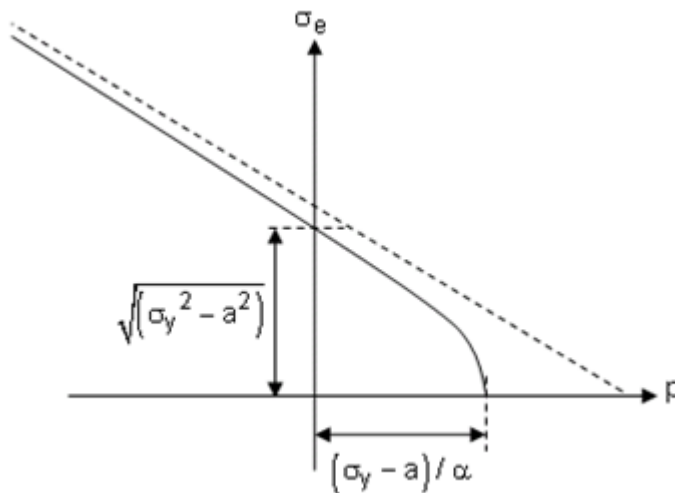
The EDP hyperbolic yield criteria form is:

$$f(\sigma, \sigma_y) = \sqrt{a^2 + \sigma_e^2} + \alpha \frac{1}{3} \text{tr}(\sigma) - \sigma_y = 0$$

where the constant a , pressure sensitivity α , and uniaxial yield stress σ_y are the user-defined parameters.

In the following figure, the hyperbolic yield criterion is plotted and compared to the linear yield criterion shown in the dashed line:

Figure 4.13: Hyperbolic and Linear Criterion in the Meridian Plane



Defining the EDP Hyperbolic Yield Criterion

After initializing the extended Drucker-Prager hyperbolic yield criterion (**TB,EDP,,,,HYFUN**), enter the following constants (**TBDATA**):

Constant	Meaning	Property
C1	α	Pressure sensitivity
C2	a	Material parameter
C3	σ_y	Uniaxial yield stress

The constants can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

4.4.4.2.2. EDP Plastic Flow Potentials

Three EDP flow potentials correspond in form to each of the [yield criteria](#) (p. 55). However, the user-defined parameters for the flow potentials are independent of those for the yield criteria, and any potential can be combined with any yield criterion.

The EDP plastic flow potentials include the following forms:

4.4.4.2.2.1. Linear Form

4.4.4.2.2.2. Power Law Form

4.4.4.2.2.3. Hyperbolic Form

4.4.4.2.2.1. Linear Form

The linear form of the plastic flow potential is:

$$Q(\sigma, \sigma_y) = \sigma_e + \bar{\alpha} \frac{1}{3} \text{tr}(\sigma)$$

where $\bar{\alpha}$ is the flow potential pressure sensitivity.

Defining the Linear Plastic Flow Potential

After initializing the material data table (**TB**,EDP,,,,LFPOT), enter the following constant (**TB**DATA):

Constant	Meaning	Property
C1	$\bar{\alpha}$	Pressure sensitivity

The material behavior can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

4.4.4.2.2.2. Power Law Form

The power law form of the plastic flow potential is:

$$Q(\sigma, \sigma_y) = \sigma_e^{\bar{b}} + \bar{\alpha} \frac{1}{3} \text{tr}(\sigma)$$

where the exponent $\bar{b}$ and the pressure sensitivity $\bar{\alpha}$ are user-defined parameters.

Defining the Linear Plastic Flow Potential

After initializing the material data table (**TB**,EDP,,,,PFPOT), enter the following constants (**TB**DATA):

Constant	Meaning	Property
C1	$\bar{\alpha}$	Pressure sensitivity
C2	$\bar{b}$	Exponent

The material behavior can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

4.4.4.2.2.3. Hyperbolic Form

The hyperbolic form of the plastic flow potential is:

$$Q(\sigma, \sigma_y) = \sqrt{\bar{a}^2 + \sigma_e^2} + \bar{a} \frac{1}{3} \text{tr}(\sigma)$$

where the pressure sensitivity $\bar{a}$ the constant $\bar{a}$ are user-defined parameters.

Defining the Linear Plastic Flow Potential

After initializing the material data table (**TB**,EDP,,,,HFOT), enter the following constants (**TB-DATA**):

Constant	Meaning	Property
C1	$\bar{a}$	Pressure sensitivity
C2	$\bar{a}$	Material parameter

The material behavior can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

4.4.4.2.3. Plastic Strain Increments for Flow Potentials

The plastic strain increment corresponding to each of the plastic flow potentials (p. 58) is:

$$\begin{aligned} d\epsilon^{pl} &= d\lambda \left[\sqrt{\frac{3}{2}} \mathbf{s} + \frac{\bar{a}}{3} \mathbf{I} \right] \\ d\epsilon^{pl} &= d\lambda \left[\sqrt{\frac{3}{2}} \mathbf{s} (\bar{b} q^{\bar{b}-1}) + \frac{\bar{a}}{3} \mathbf{I} \right] \\ d\epsilon^{pl} &= d\lambda \left[\sqrt{\frac{3}{2}} \mathbf{s} \left(\frac{q}{\sqrt{\bar{a}^2 + q^2}} \right) + \frac{\bar{a}}{3} \mathbf{I} \right] \end{aligned}$$

where $\mathbf{s}$ is the deviatoric stress:

$$\mathbf{s} = \sigma - \frac{1}{3} \text{tr}(\sigma) \mathbf{I}$$

The dilatation for each of the flow potentials is:

$$\text{tr}(d\epsilon^{pl}) = \bar{a} d\lambda$$

Associated flow is obtained if the plastic potential form and parameters are set equal to the yield criterion.

4.4.4.2.4. Example EDP Material Model Definitions

The following examples show how to define the EDP material model using various yield criteria and flow potentials:

Example 4.14: EDP -- Linear Yield Criterion and Flow Potential

```
/prep7
!!! Define linear elasticity constants
mp,ex,1,2.1e4
mp,nuxy,1,0.45
```

```

! Extended Drucker-Prager Material Model Definition

! Linear Yield Function
tb,edp,1,1,2,LYFUN
tbdata,1,2.2526,7.894657

! Linear Plastic Flow Potential
tb,edp,1,1,2,LFPOT
tbdata,1,0.566206

```

Example 4.15: EDP -- Power Law Yield Criterion and Flow Potential

```

/prep7
!!! Define linear elasticity constants
mp,ex,1,2.1e4
mp,nuxy,1,0.45

! Extended Drucker-Prager Material Model Definition

! Power Law Yield Function
tb,edp,1,1,3,PYFUN
tbdata,1,8.33,1.5

! Power Law Plastic Flow Potential
tb,edp,1,1,2,PFPOT
tbdata,1,8.33,1.5

```

Example 4.16: EDP -- Hyperbolic Yield Criterion and Flow Potential

```

/prep7
!!! Define linear elasticity constants
mp,ex,1,2.1e4
mp,nuxy,1,0.45

! Extended Drucker-Prager Material Model Definition

! Hyperbolic Yield Function
tb,edp,1,1,3,HYFUN
tbdata,1,1.0,1.75,7.89

! Hyperbolic Plastic Flow Potential
tb,edp,1,1,2,HFPOT
tbdata,1,1.0,1.75

```

4.4.4.3. Extended Drucker-Prager Cap

The EDP Cap material model has a yield criterion similar to the other [extended Drucker-Prager yield criteria](#) (p. 55) with the addition of two cap surfaces that truncate the yield surface in tension and compression regions [11]. The model formulation follows that of Schwer and Murray [12] and Foster et al [13] and the numerical formulation is modified from the work of Pelessone [14].

The criterion is a function of the three stress invariants I_1 , J_2 , and J_3 , given by:

$$\begin{aligned}
 I_1 &= \text{trace}(\boldsymbol{\sigma}) \\
 J_2 &= \frac{1}{2} \mathbf{s} : \mathbf{s} \\
 J_3 &= \det(\mathbf{s})
 \end{aligned}$$

where $\mathbf{s}$ is the deviatoric stress.

Three functions define the surfaces that make up the yield criterion. The shear envelope function is given by:

$$Y_s(I_1, \sigma_c) = \sigma_c - A \exp(BI_1) - \alpha I_1$$

where σ_c is the cohesion related yield parameter and is a user defined material parameter along with A , B , and α . This function reduces to the Drucker-Prager criterion for $A=0$. For positive values of I_1 , the shear failure envelope is evaluated at $I_1 = 0$, which gives the constant value $(Y_s(0, \sigma_c) = \sigma_c - A)$.

The compaction function is itself a function of the shear envelope function and is given by:

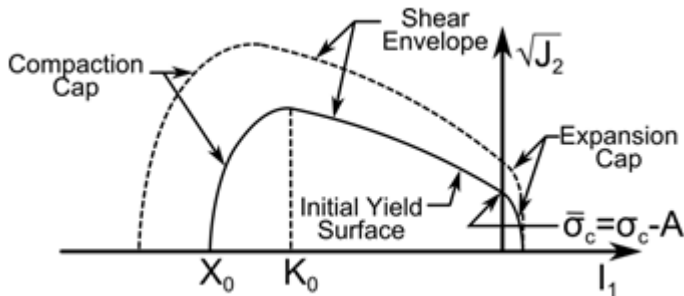
$$Y_c(I_1, K_0, \sigma_c) = 1 - H(K_0 - I_1) \left(\frac{I_1 - K_0}{R_c Y_s(K_0, \sigma_c)} \right)^2$$

where H is the Heaviside step function, R_c is a user-input material parameter, and K_0 defines the intersection of the compaction surface with the shear envelope, given by:

$$K_0 = X_0 + R_c Y_s(K_0, \sigma_c)$$

where X_0 is the user-defined value of I_1 at the intersection of the compaction cap with $J_2=0$, as shown in the following figure:

Figure 4.14: Yield Surface for the Cap Criterion



The compaction function defines the material yield surface when $I_1 < K_0$.

The expansion function is a function of the shear envelope function and is given by:

$$Y_t(I_1, \sigma_c) = 1 - H(I_1) \left(\frac{I_1}{R_t Y_s(0, \sigma_c)} \right)^2$$

where R_t is a user-input material parameter. The expansion function defines the material yield surface when $I_1 > 0$. The expansion cap function reaches peak value at $I_1 = R_t Y_s(0, \sigma_c)$.

These functions define the yield criterion, given by:

$$f(I_1, J_2, J_3, K_0, \sigma_c) = \Gamma^2 J_2 - Y_c(I_1, K_0, \sigma_c) Y_t(I_1, \sigma_c) Y_s^2(I_1, \sigma_c) \quad (4.10)$$

where Γ is the Lode angle function. The Lode angle is given by:

$$\beta(J_2, J_3) = -\frac{1}{3} \sin^{-1} \left(\frac{3\sqrt{3} J_3}{2J_2^{3/2}} \right)$$

and the Lode angle function is:

$$\Gamma = \frac{1}{2} \left(1 + \sin(3\beta) + \frac{1}{\psi} (1 - \sin(3\beta)) \right)$$

where ψ is a user-defined material parameter, a ratio of the extension strength to compression strength in triaxial loading.

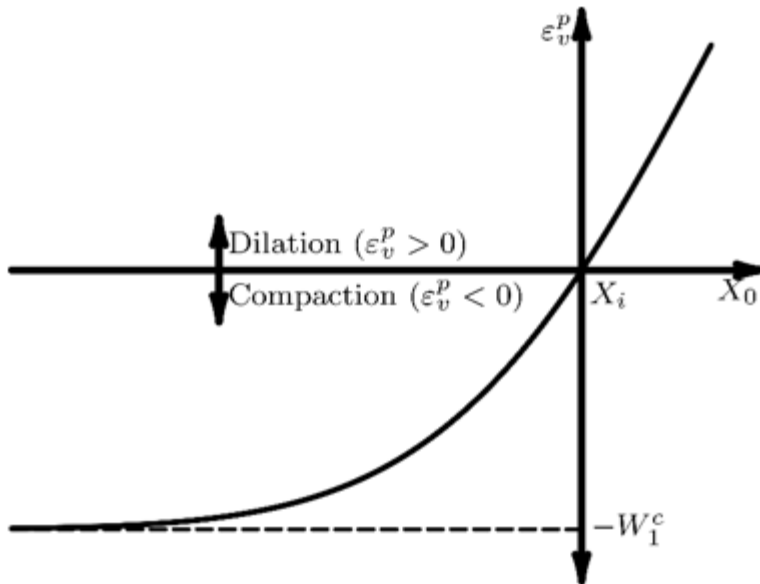
Two methods can be used to evolve the yield criterion due to plastic deformation. Hardening or softening of the compaction cap is due to evolution of X_0 , which is the intersection of the cap surface with $J_2=0$ shown in [Figure 4.14: Yield Surface for the Cap Criterion \(p. 61\)](#). This value evolves due to plastic volume strain ε_v^p , and the relationship is given by [15]:

$$\varepsilon_v^p = W_1^c \left[\exp \left((D_1^c - D_2^c (X_0 - X_i)) (X_0 - X_i) \right) - 1 \right]$$

where X_i is the initial value of X_0 , and the user-defined parameters are W_1^c , D_1^c , and D_2^c . Shown in [Figure 4.15: Relationship Between \$\varepsilon_v^p\$ and \$X_0\$ \(p. 62\)](#), evolution of X_0 includes both hardening and softening depending on the volumetric plastic strain. Plastic dilation can occur on the shear envelope, causing the magnitude of X_0 to decrease; in this case, to ensure a monotonic relationship between ε_v^p and X_0 , the parameters must satisfy:

$$D_1^c > 2D_2^c (X_0 - X_i)$$

Figure 4.15: Relationship Between ε_v^p and X_0



Evolution of the yield surface at the intersection of the shear envelope with the expansion cap occurs by combining the cap model with an [isotropic hardening model \(p. 36\)](#) to evolve the value of $\bar{\sigma}_c$. The bilinear, multilinear, or nonlinear isotropic hardening function can be used, and the yield stress

from the isotropic hardening model must be consistent with the value of $\bar{\sigma}_c$ calculated from the cap material parameters given by $\bar{\sigma}_c = \sigma_c - A$.

The plastic potential used to determine the direction of the plastic strain increment has the same form as the yield criterion given by [Equation 4.10 \(p. 61\)](#):

$$Q(I_1, J_2, J_3, K_0, \sigma_c) = \Gamma^2 J_2 - \bar{Y}_c(I_1, K_0, \sigma_c) \bar{Y}_t(I_1, \sigma_c) \bar{Y}_s^2(I_1, \sigma_c) \quad (4.11)$$

where the potential surface functions are:

$$\begin{aligned} \bar{Y}_s(I_1, \sigma_c) &= \sigma_c - A \exp(\bar{B} I_1) - \bar{\alpha} I_1 \\ \bar{Y}_c(I_1, K_0, \sigma_c) &= 1 - H(K_0 - I_1) \left(\frac{I_1 - K_0}{\bar{R}_c \bar{Y}_s(K_0, \sigma_c)} \right)^2 \\ \bar{Y}_t(I_1, \sigma_c) &= 1 - H(I_1) \left(\frac{I_1}{\bar{R}_t \bar{Y}_s(0, \sigma_c)} \right)^2 \end{aligned}$$

and the user defined potential parameters are $\bar{B}$, $\bar{\alpha}$, $\bar{R}_c$ and $\bar{R}_t$.

The following topics for defining the EDP Cap material model are available:

[4.4.4.3.1. Defining the EDP Cap Yield Criterion and Hardening](#)

[4.4.4.3.2. Defining the EDP Cap Plastic Potential](#)

[4.4.4.3.3. Example EDP Cap Material Model Definition](#)

4.4.4.3.1. Defining the EDP Cap Yield Criterion and Hardening

After initializing the material data table (**TB**,EDP,,,CYFUN), enter the following constants (**TB**DATA):

Constant	Meaning	Property
C1	R_c	Compaction cap parameter
C2	R_t	Expansion cap parameter
C3	X_i	Compaction cap yield pressure
C4	σ_c	Cohesion yield parameter
C5	B	Shear envelope exponent
C6	A	Shear envelope exponential coefficient
C7	α	Shear envelope linear coefficient
C8	ψ	Ratio of extension to compression strength
C9	W_1^c	Limiting value of volumetric plastic strain
C10	D_1^c	Hardening parameter
C11	D_2^c	Hardening parameter

The yield criterion and hardening behavior can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

4.4.4.3.2. Defining the EDP Cap Plastic Potential

After initializing the material data table (**TB,EDP,,,CFPOT**), enter the following constants (**TB,DATA**):

Constant	Meaning	Property
C1	$\bar{R}_c$	Compaction cap parameter
C2	$\bar{R}_t$	Expansion cap parameter
C3	$\bar{B}$	Shear envelope exponent
C4	$\bar{\alpha}$	Shear envelope linear coefficient

The plastic flow potential can be defined as a function of temperature (*NTEMP* on the **TB** command), with temperatures specified for the table entries (**TBTEMP**).

If the plastic flow potential is not defined, the yield surface is used as the flow potential, resulting in an associated flow model.

4.4.4.3.3. Example EDP Cap Material Model Definition

The following example input shows how to define an EDP Cap model by defining the yield criterion, hardening, and plastic flow potential:

Example 4.17: EDP Cap Model Material Constant Input

```
/prep7
! Define linear elasticity constants
mp,ex,1,14e3
mp,nuxy,1,0.0

! Cap yield function
tb,edp,1,1,,cyfun
tbdata,1,2          ! Rc
tbdata,2,1.5        ! Rt
tbdata,3,-80        ! Xi
tbdata,4,10         ! SIGMA
tbdata,5,0.001      ! B
tbdata,6,2          ! A
tbdata,7,0.05       ! ALPHA
tbdata,8,0.9        ! PSI

! Define hardening for cap-compaction portion
tbdata,9,0.6        ! W1c
tbdata,10,3.0/1000  ! D1c
tbdata,11,0.0       ! D2c

! Cap plastic flow potential function
tb,edp,1,1,,cfpot
tbdata,1,2          ! RC
tbdata,2,1.5        ! RT
tbdata,3,0.001      ! B
tbdata,4,0.05       ! ALPHA
```

4.4.5. Gurson

Use the Gurson model to represent plasticity and damage in ductile porous metals [16][17]. When plasticity and damage occur, ductile metal undergoes a process of void growth, nucleation, and coalescence. The model incorporates these microscopic material behaviors into macroscopic plasticity behavior based on changes in the void volume fraction, also known as porosity, and pressure. A

porosity increase corresponds to an increase in material damage, resulting in a diminished load-carrying capacity.

The yield criterion and flow potential for the Gurson model is:

$$f(\sigma, \sigma_y) = \left(\frac{\sigma_e}{\sigma_y} \right)^2 + 2f^* q_1 \cosh \left(\frac{3}{2} \frac{q_2 p}{\sigma_y} \right) - (1 + q_3 f^{*2}) = 0$$

where σ_e is the von Mises equivalent stress, σ_y is the yield stress, q_1 , q_2 , and q_3 are user-input Tvergaard-Needleman constants, and f^* is the modified void volume fraction. The hydrostatic pressure p is defined as:

$$p = \frac{1}{3} \text{trace}(\sigma)$$

The following additional Gurson model topics are available:

[4.4.5.1. Void Volume Fraction](#)

[4.4.5.2. Hardening](#)

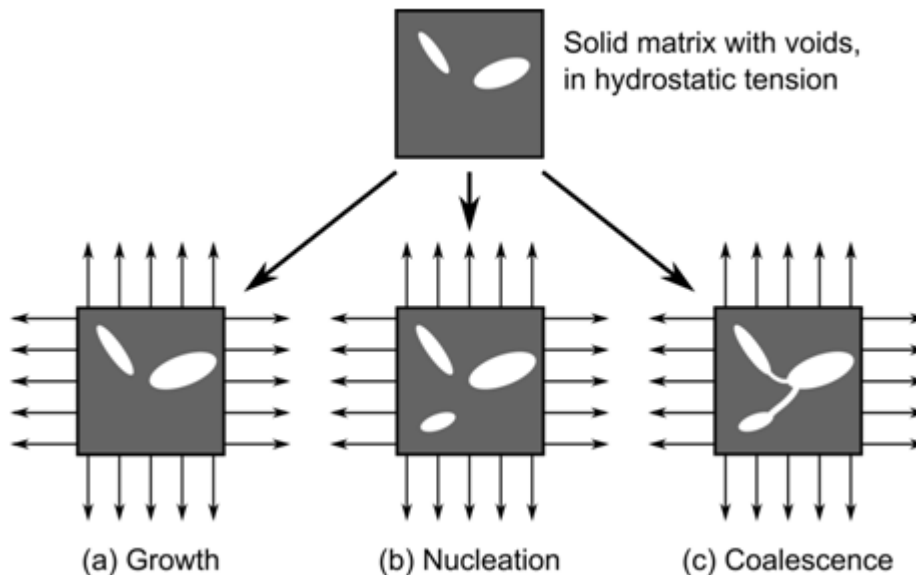
[4.4.5.3. Defining the Gurson Material Model](#)

For more information, see [Gurson's Model in the Theory Reference](#).

4.4.5.1. Void Volume Fraction

The following figure shows the phenomena of voids at the microscopic scale that are incorporated into the Gurson model:

Figure 4.16: Growth, Nucleation, and Coalescence of Voids at Microscopic Scale



(a): Existing voids grow when the solid matrix is in a hydrostatic-tension state. The solid matrix is assumed to be incompressible in plasticity so that any material volume growth is due to the void volume expansion.

(b): Void nucleation occurs, where new voids are created during plastic deformation due to debonding of the inclusion-matrix or particle-matrix interface, or from the fracture of the inclusions or particles themselves.

(c): Voids coalesce. In this process, the isolated voids establish connections. Although coalescence may not discernibly affect the void volume, the load-carrying capacity of the material begins to decay more rapidly at this stage.

The void volume fraction is the ratio of void volume to the total volume. A volume fraction of 0 indicates no voids and the yield criterion reduces to the von Mises criterion. A volume fraction of 1 indicates all the material is void. The initial void volume fraction, f_0 , is a user-defined parameter, and the rate of change of void volume fraction is a combination of the rate of growth and the rate of nucleation:

$$\dot{f} = \dot{f}_{growth} + \dot{f}_{nucleation}$$

From the assumption of isochoric plasticity and conservation of mass, the rate of change of void volume fraction due to growth is proportional to the rate of volumetric plastic strain:

$$\dot{f}_{growth} = (1-f) \dot{\epsilon}^{pl} : I$$

Void nucleation is controlled by either the plastic strain or the stress, and is assumed to follow a normal distribution of statistics.

In the case of strain-controlled nucleation, the distribution is described by the mean strain, $\bar{\epsilon}_N$, and deviation, S_N . The void nucleation rate due to strain control is given by:

$$\dot{f}_{nucleation} = \frac{f_N \dot{\epsilon}^{pl}}{S_N \sqrt{2\pi}} e^{-\frac{1}{2} \left(\frac{\bar{\epsilon}^{pl} - \bar{\epsilon}_N}{S_N} \right)^2}$$

where f_N is the maximum void fraction for nucleated voids, $\bar{\epsilon}^{pl}$ is the effective plastic strain, and the rate of effective plastic strain, $\dot{\epsilon}^{pl}$, is determined by equating the microscopic plastic work to the macroscopic plastic work:

$$(1-f) \sigma_y \dot{\epsilon}^{pl} = \sigma : \dot{\epsilon}$$

In the case of stress-controlled nucleation, void nucleation is determined by the distribution of maximum normal stress on the interfaces between inclusions and the matrix, equal to $\sigma_y + p$. Stress-controlled nucleation takes into account the effect of triaxial loading on the rate of void nucleation. The void-nucleation rate for stress control is given by:

$$\dot{f}_{nucleation} = \frac{f_N (\dot{\sigma}_y + \dot{p})}{S_N^\sigma \sqrt{2\pi}} e^{-\frac{1}{2} \left(\frac{\sigma_y + p - \sigma_N}{S_N^\sigma} \right)^2}$$

where distribution of stress is described by the mean stress, σ_N and deviation, S_N^σ .

The modified void volume fraction, f^* , is used to model the loss of material load carrying capacity associated with void coalescence. When the current void volume fraction reaches a critical value f_c , the material load carrying capacity decreases rapidly due to coalescence. When the void volume fraction reaches f_F , the load-carrying capacity of the material is lost completely. The modified void volume fraction is given by:

$$f^* = \begin{cases} f & \text{if } f \leq f_c \\ f_c + \frac{1}{f_F - f_c} (f - f_c) & \text{if } f > f_c \end{cases}$$

4.4.5.2. Hardening

The Gurson model can be combined with [one of the isotropic hardening models \(p. 40\)](#) to incorporate isotropic hardening of the yield stress in the Gurson yield criterion.

To combine the Gurson model with [Chaboche nonlinear kinematic hardening \(p. 51\) \[22\]](#), the yield criterion is modified to:

$$f(\sigma, \sigma_y) = \left(\frac{\bar{\sigma}_e}{\sigma_y} \right)^2 + 2f^* q_1 \cosh \left(\frac{3}{2} \frac{q_2 \bar{p}}{\sigma_y} \right) - (1 + q_3 f^{*2}) = 0$$

where $\bar{\sigma}_e$ is the von Mises equivalent modified relative stress, and $\bar{p}$ is the effective hydrostatic pressure defined as:

$$\bar{p} = \frac{1}{3} \text{trace}(\bar{\sigma})$$

where $\bar{\sigma}$ is the modified relative stress, which itself is a function of the modified backstress α_{eff} :

$$\bar{\sigma} = \sigma - \alpha_{eff}$$

$$\alpha_{eff} = (1 - \sqrt{q_3} f^*) \alpha$$

where α is the kinematic hardening backstress.

The rate of effective plastic strain equation is also modified to:

$$(1 - f) \sigma_y \dot{\varepsilon}^{pl} = \dot{\sigma} : \varepsilon^p$$

For more information, see [Gurson Plasticity with Isotropic/Chaboche Kinematic Hardening in the Theory Reference](#).

4.4.5.3. Defining the Gurson Material Model

The Gurson material model requires material parameters for the base model combined with parameters for either strain-controlled or stress-controlled nucleation. Additional input is required to define the void coalescence behavior.

4.4.5.3.1. Defining the Gurson Base Model

4.4.5.3.2. Defining Stress- or Strain-Controlled Nucleation

4.4.5.3.3. Defining the Void Coalescence Behavior

4.4.5.3.4. Example Gurson Model Definition

4.4.5.3.1. Defining the Gurson Base Model

To define the Gurson base model, initialize the material data table (**TB,GURSON,,,,BASE**), then input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	σ_y	Initial yield strength
C2	f_0	Initial porosity
C3	q_1	First Tvergaard-Needleman constant
C4	q_2	Second Tvergaard-Needleman constant
C5	q_3	Third Tvergaard-Needleman constant

4.4.5.3.2. Defining Stress- or Strain-Controlled Nucleation

The Gurson base model (p. 68) is combined with either stress- or strain-controlled nucleation.

To define stress-controlled nucleation, initialize the material data table (**TB,GURSON,,,,SSNU**), then input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	f_N	Nucleation porosity
C2	σ_N	Mean stress
C3	S_N^σ	Stress standard deviation

To define strain-controlled nucleation, initialize the material data table (**TB,GURSON,,,,SNNU**), then input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	f_N	Nucleation porosity
C2	ϵ_N	Mean strain
C3	S_N	Strain standard deviation

4.4.5.3.3. Defining the Void Coalescence Behavior

Define the void coalescence behavior after defining the Gurson base model (p. 68) and either the stress- or strain-controlled nucleation (p. 68) behavior.

Initialize the material data table (**TB,GURSON,,,,COAL**), then input the following constants (**TBDATA**):

Constant	Meaning	Property
C1	f_c	Critical porosity
C2	f_F	Failure porosity

4.4.5.3.4. Example Gurson Model Definition

Following is an example Gurson plasticity material model definition:

Example 4.18: Gurson Model with Isotropic and Kinematic Hardening

```

/prep7
!!! Define linear elasticity constants
mp,ex,1,207.4E3      ! Young modulus (MPa)
mp,nuxy,1,0.3        ! Poisson's ratio

!!! Define parameters related to Gurson model with
!!! the option of strain-controlled nucleation with
!!! coalescence

f_0=1E-8             ! initial porosity
q1=1.5               ! first Tvergaard constant
q2=1.0               ! second Tvergaard constant
q3=2.25              ! third Tvergaard constant = q1^2
f_c=0.15             ! critical porosity
f_F=0.25             ! failure porosity
f_N=0.04             ! nucleation porosity
s_N=0.1              ! standard deviation of mean strain
strain_N=0.3         ! mean strain
sigma_Y=755          ! initial yielding strength (MPa)
power_N=0.1          ! power value for nonlinear isotropic
                    ! hardening power law

!base model
tb,gurson,1,,5,base
tbdata,1,sigma_Y,f_0,q1,q2,q3

! Strain-controlled nucleation
tb,gurson,1,,3,snnu
tbdata,1,f_N,strain_N,s_N

! Coalescence
tb,gurson,1,,2,coal
tbdata,1,f_c,f_F

! Power law isotropic hardening
tb,nliso,1,,2,POWER
tbdata,1,sigma_Y,power_N

```

4.4.6. Cast Iron

The cast iron plasticity model is used to model gray cast iron. The microstructure of gray cast iron is a two-phase material with graphite flakes embedded in a steel matrix [20]. The microstructure leads to different behavior in tension and compression. In tension, cracks form due to the graphite flakes and the material is brittle with low strength. In compression, the graphite flakes behave as incompressible media that transmit stress and the steel matrix governs the overall behavior.

The model is isotropic elastic with the same elastic behavior in tension and compression. The yield strength and isotropic hardening behavior may be different in tension and in compression. Different yield criteria and plastic flow potentials are used for tension and compression. It can be combined with [Chaboche nonlinear kinematic hardening \(p. 51\)](#) to simulate cast iron behavior in cyclic loading.

A composite yield surface is used to model different yield behavior in tension and compression. The tension behavior is pressure-dependent and the Rankine maximum stress criterion is used:

$$f_t(\sigma, \sigma_y^t) = \frac{2}{3} \cos(\beta) \sigma_e + p - \sigma_y^t = 0$$

where σ_y^t is the uniaxial tension yield stress. The hydrostatic pressure p , the von Mises equivalent stress σ_e , and the Lode angle β are defined as:

$$p = \frac{1}{3} \text{trace}(\sigma)$$

$$\sigma_e = \left(\frac{3}{2} \bar{\mathbf{s}} : \bar{\mathbf{s}} \right)^{\frac{1}{2}}$$

$$\beta = \frac{1}{3} \cos^{-1} \left(\frac{3\sqrt{3} J_3}{2 J_2^{3/2}} \right)$$

where J_2 and J_3 are the stress invariants:

$$J_2 = \frac{1}{2} \bar{\mathbf{s}} : \bar{\mathbf{s}}$$

$$J_3 = \det(\bar{\mathbf{s}})$$

where $\bar{\mathbf{s}}$ is the relative deviatoric stress.

In compression, the pressure-independent [von Mises \(p. 37\)](#) yield criterion is used:

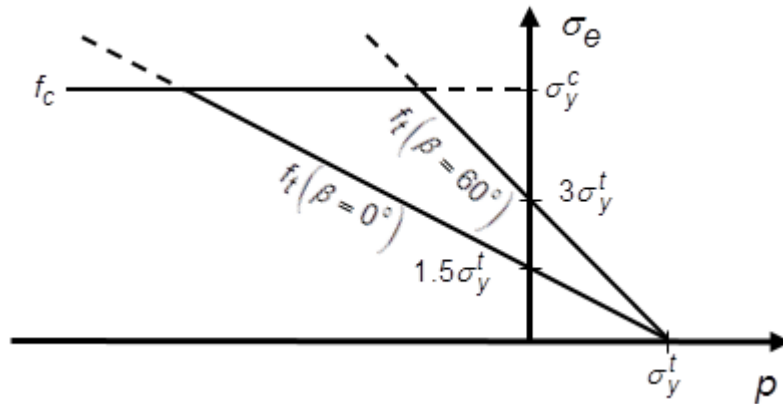
$$f_c(\sigma, \sigma_y^c) = \sigma_e - \sigma_y^c = 0$$

The relative deviatoric stress is:

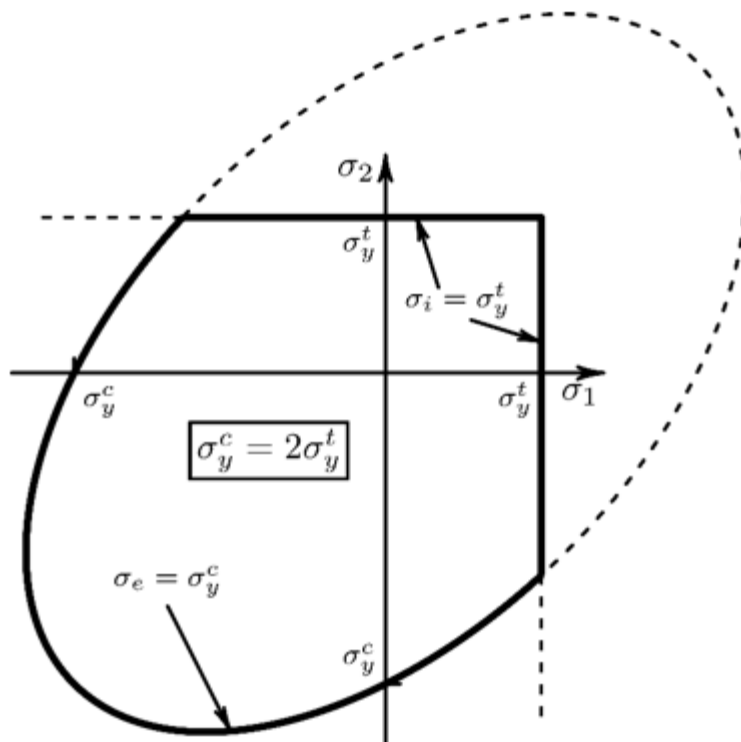
$$\bar{\mathbf{s}} = \sigma - p\mathbf{I} - \alpha$$

where α is the nonlinear kinematic backstress given by [Equation 4.9 \(p. 52\)](#) and [Equation 4.8 \(p. 52\)](#).

The following figure shows the composite yield surface in the meridional plane in which the ordinate and abscissa are von Mises equivalent stress and pressure, respectively:

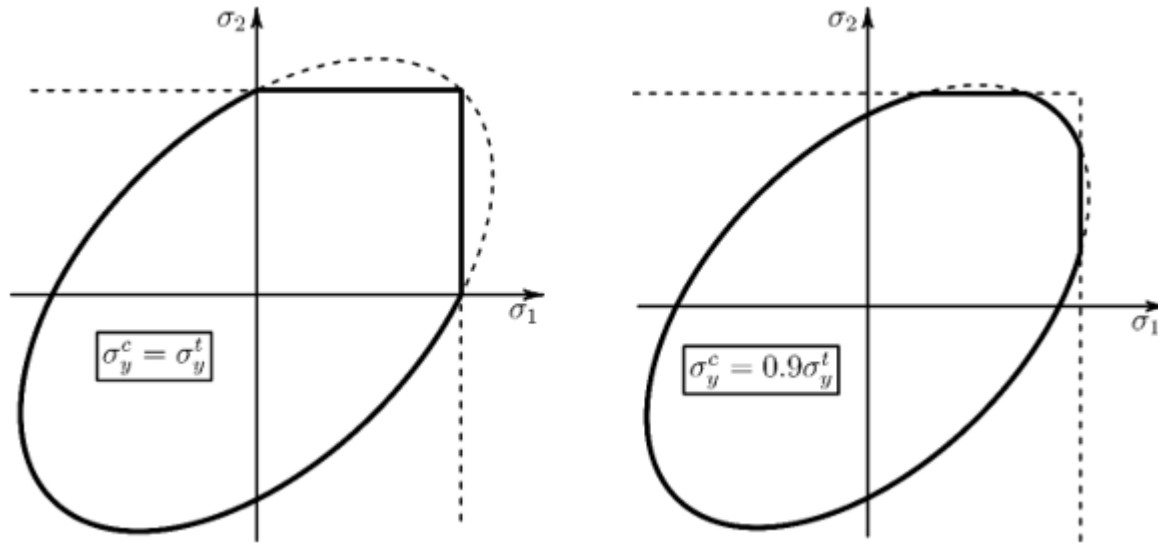
Figure 4.17: Cast Iron Yield Surface

In the following figure, the solid line shows the composite cast iron yield surface in 2-D principal stress space:

Figure 4.18: Composite Cast Iron Yield Surface in 2-D Principal Stress

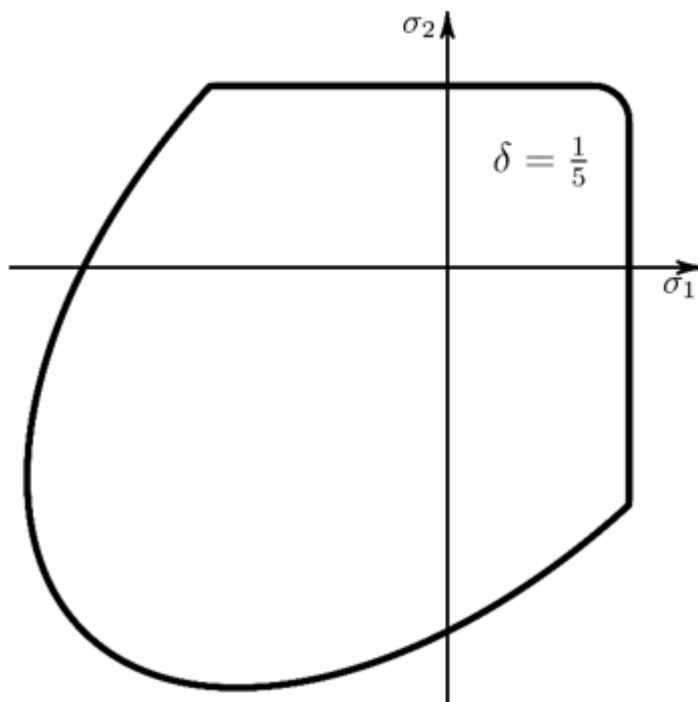
The figure shows that for biaxial loading, the maximum principal stress cannot exceed the tension yield stress. For some combinations of tension and compression loading, the material can yield on either the tension yield surface or the compression yield surface.

As shown in this figure, when the tension and compression yield stress are equal, uniaxial loading yields at the intersection of the compression and tension yield surfaces:

Figure 4.19: Comparison of Cast Iron Yield Surfaces

When the tension yield stress is greater than the compression yield stress, uniaxial loading yields on the compression surface. Because this is not the intended behavior of the cast iron model, ANSYS, Inc. recommends that the material definition give $\sigma_y^c > \sigma_y^t$ for all states of material evolution.

As shown in the following figure, the tension surface can include a rounding factor, δ , to smooth the sharp corners and improve convergence.

Figure 4.20: Cast Iron Yield Surface with Tension Rounding

The default is no rounding, $\delta=0$, and the maximum is $\delta=1$.

When a rounding factor is used, the tension yield criterion becomes:

$$f_t(\sigma, \sigma_y^t) = -\sigma_y^t + \begin{cases} 0 & \bar{s}_3 \leq 0 \\ \frac{\bar{s}_3^2}{\delta^2 \sigma_y^t} & \bar{s}_2 \leq 0 \text{ and } \bar{s}_3 > 0 \\ \frac{\bar{s}_2^2 + \bar{s}_3^2}{\delta^2 \sigma_y^t} & \bar{s}_1 \leq 0 \text{ and } \bar{s}_2 > 0 \\ \frac{\bar{s}_1^2 + \bar{s}_2^2 + \bar{s}_3^2}{\delta^2 \sigma_y^t} & \bar{s}_1 > 0 \end{cases}$$

where $\bar{s}_i = \bar{s}_i - (1-\delta)\sigma_y^t$ and $\bar{s}_3 \geq \bar{s}_2 \geq \bar{s}_1$ are the ordered principal relative stress values.

The evolution of the yield stress in tension and compression follows the piecewise user-defined linear stress-strain curves for compression and tension. The tension yield stress evolves as a function of the equivalent uniaxial plastic strain, $\bar{\epsilon}_t^{pl}$. The evolution of the equivalent uniaxial plastic strain is defined by equating the uniaxial plastic work increment to the total plastic work increment:

$$\sigma_y^t d\bar{\epsilon}_t^{pl} = \sigma : d\epsilon^{pl}$$

The compression yield stress evolves as a function of the equivalent plastic strain, $\bar{\epsilon}^{pl}$, which is calculated from the increment in plastic strain determined by consistency with the yield criterion and the flow potential.

The plastic flow potential is defined by the von Mises yield criterion in compression and results in an associated flow rule. The flow potential in compression is:

$$Q_c = \frac{\sigma_e}{3} \quad \text{for } p < -\frac{\sigma_y^c}{3}$$

In tension, the Rankine cap yield surface is replaced by an ellipsoidal surface defined by:

$$\frac{(p + Q_t)^2}{c^2} + \sigma_e^2 = 9Q_t^2 \quad \text{for } p \geq -\frac{\sigma_y^c}{3}$$

where c is a constant function of the user-defined plastic Poisson's ratio, ν^{pl} :

$$c = \sqrt{\frac{5 + 2\nu^{pl}}{9(1 - 2\nu^{pl})}}$$

The plastic Poisson's ratio is the absolute value of the ratio of the transverse to the longitudinal plastic strain under uniaxial tension. It determines the amount of volumetric expansion during tensile plastic deformation. For $\nu^{pl} = 0.5$, the tensile flow potential reduces to the von Mises potential. The tensile flow potential gives a nonassociated flow model and results in an unsymmetric material stiffness tensor.

When the cast iron model is combined with nonlinear kinematic hardening, the backstress is always deviatoric. In tension, only the deviatoric part of the plastic strain is used in the backstress evolution equation.

4.4.6.1. Defining the Cast Iron Material Model

Define the isotropic elastic behavior (**MP** or **TB,ELASTIC**). Initialize the material data table (**TB,CAST,,,,ISOTROPIC**) and input the following constant:

Constant	Meaning	Property
C1	ν^{pl}	Plastic Poisson's ratio

Enter the tension yield surface rounding factor by initializing the material data table (**TB,CAST,,,,ROUNDING**) and input the following constant:

Constant	Meaning	Property
C1	δ	Tension yield surface rounding factor

Enter the tensile multilinear hardening stress-strain points into a data table (**TB,CAST,,,,TENSION**). Do the same for the compressive multilinear hardening stress-strain points (**TB,CAST,,,,COMPRESSION**). Enter the tension and compression stress-strain points into their respective tables via the **TBPT** command, with the compression points being entered as positive values:

Constant	Meaning	Property
X	ε_i	Plastic strain value
Y	σ_i	Stress value

The plastic Poisson's ratio and stress-strain points can be defined as a function of temperature (**NTEMP** value on the **TB** command), with individual temperatures specified for the table entries (**TBTEMP**).

Example 4.19: Cast Iron

```
/prep7
mp, ex, 1,14.773E6
mp, nuxy, 1,0.2273

! Define cast iron model
TB,CAST,1,,,ISOTROPIC
TB,DATA,1,0.04

TB,CAST,1,1,5,TENSION
TBPT,,0.000E-00,0.813E+04
TBPT,,1.13E-04,0.131E+05
TBPT,,8.69E-04,0.241E+05
TBPT,,1.55E-03,0.288E+05
TBPT,,2.32E-03,0.322E+05

TB,CAST,1,1,5,COMPRESSION
TBPT,,0.000E-00,0.300E+05
TBPT,,1.62E-03,0.500E+05
TBPT,,4.07E-03,0.581E+05
TBPT,,6.56E-03,0.656E+05
TBPT,,9.26E-03,0.700E+05

TB,CHABOCHE,1,1,3,TRATE
TB,DATA,1,0,4.0783E3,0.8165
TB,DATA,4,3.8973E4,743.0119,3.8973E5,7430.119
```

4.4.7. Crushable Foam

Crushable foam materials are cellular solids typically made of metal or polymer. They have a small elastic range followed by nonlinear permanent deformation governed by buckling and deformation in the cell walls and struts. Common applications include packaging materials such as expanded polystyrene where energy absorption is important, and foam-core sandwich panels where a high stiffness-to-weight ratio is required.

Based on the initial work of Desphande and Fleck [1], the model captures the monotonic yielding, hardening evolution, and elastic unloading of crushable foam materials. It is likely unsuitable for high-rate loading, large deformations, or non-monotonic yield behavior such as cyclic loading.

The yield behavior of crushable foams depends on both the deviatoric and hydrostatic loading. Shown in [Figure 4.21: Crushable Foam Yield Surface \(p. 75\)](#), the yield function is:

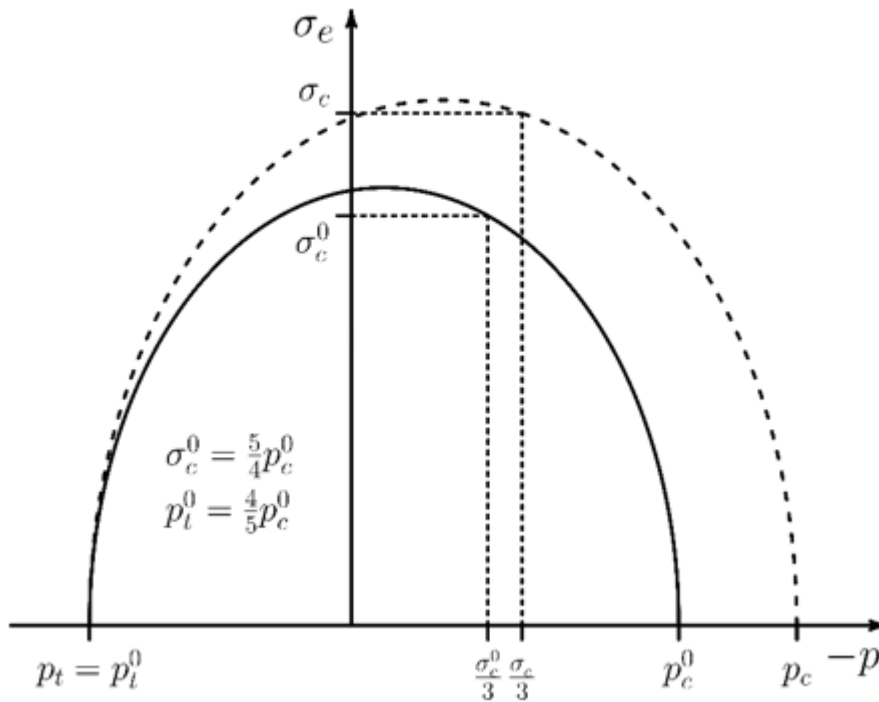
$$f(\sigma, \sigma_c, p_c) = \sqrt{\sigma_e^2 + \alpha^2(p - \bar{p})^2} - Y_s$$

where σ_c and p_c are the evolving uniaxial and absolute hydrostatic compression yield values, and the von Mises effective stress and hydrostatic pressure are:

$$\sigma_e = \left(\frac{3}{2}s:s\right)^{\frac{1}{2}}$$

$$p = \frac{1}{3}\text{trace}(\sigma)$$

Figure 4.21: Crushable Foam Yield Surface



The derived material parameters are the shape factor α , the hydrostatic center of the yield surface $\bar{p}$, and the maximum shear Y_s . They are derived from the input parameters for initial uniaxial yield

stress σ_c^0 , initial absolute hydrostatic compression yield stress p_c^0 , and hydrostatic tension yield stress p_t . Defining the yield stress ratios,

$$k_t = \frac{p_t}{p_c^0}$$

$$k_c = \frac{\sigma_c^0}{p_c^0}$$

the derived material parameters are given by:

$$\alpha = \frac{3k_c}{\sqrt{(3k_t + k_c)(3 - k_c)}}$$

$$\bar{p} = \frac{p_c - p_t}{2}$$

$$Y_s = \alpha \frac{p_c + p_t}{2}$$

The uniaxial yield stress σ_c evolves as a function of the equivalent plastic strain κ_c :

$$\sigma_c = \Omega(\kappa_c) \sigma_c^0$$

where Ω is a user-defined multilinear hardening function.

The yield surface hardens in such a manner that both the hydrostatic tension yield p_t and the aspect ratio of the yield surface ellipse α are constant during the hardening of σ_c . This evolution gives the hardening of absolute hydrostatic compression yield as a function of the uniaxial yield hardening:

$$p_c = \frac{\sigma_c \left(\sigma_c \left(\frac{1}{\alpha^2} + \frac{1}{9} \right) + \frac{p_t}{3} \right)}{p_t + \frac{\sigma_c}{3}}$$

The evolution of the plastic equivalent strain κ_c follows one of two methods that you select. The evolution is based on the work-rate conjugate to either the [flow potential](#) or the [uniaxial yield stress](#) (p. 77):

$$\dot{\kappa}_c = \sqrt{\frac{3}{2}} \frac{\sigma \dot{\epsilon}_{pl}}{Q}$$

$$\dot{\kappa}_c = \frac{\sigma \dot{\epsilon}_{pl}}{\sigma_c}$$

where $\dot{\epsilon}_{pl}$ is the plastic strain rate and Q is the flow potential:

$$Q = \sqrt{\sigma_e + \beta^2 p^2}$$

where β is a shape factor that is a function of the plastic Poisson's ratio ν^{pl} , as follows:

$$\beta = \frac{3}{\sqrt{2}} \sqrt{\frac{1 - 2\nu^{pl}}{1 + \nu^{pl}}}$$

This non-associated flow potential results in an unsymmetric material stiffness matrix.

4.4.7.1. Defining the Crushable Foam Material Model

Define the isotropic or anisotropic elastic behavior (**MP** or **TB,ELASTIC**). Initialize the crushable foam yield data table (**TB,CFOAM,,,,YIELD**) and input the following constants:

Constant	Meaning	Property
C1	σ_c^0	Initial uniaxial compression yield
C2	p_c^0	Initial hydrostatic compression yield
C3	p_t	Hydrostatic tension yield

The yield stress values must satisfy $\sigma_c^0 > 0$, $p_c^0 > 0$, $p_t > 0$, and $p_c^0 > \sigma_c^0 / 3$.

To define the hardening of the uniaxial compressive yield, initialize a multilinear hardening data table (**TB,CFOAM,,,,MHARD**) and define the following data pairs:

Constant	Meaning	Property
X	κ_c	Plastic strain value
Y	Ω	Relative stress value

At $\kappa_c=0$, the relative stress value should start at $\Omega=1$ and increase monotonically. (Decreasing values of relative stress give softening material behavior and may lead to convergence difficulty.)

The hardening type is specified via a data table (**TB,CFOAM,,,,HTYPE**) with the following input:

Constant	Meaning	Property
C1	--	Hardening type

C1 = 0 (default) specifies that the equivalent plastic strain evolves based on work conjugate to the plastic potential. C1 = 1 specifies that the equivalent plastic strain evolves based on work conjugate to the uniaxial compressive yield stress.

The shape of the plastic potential is determined by the plastic Poisson's ratio, entered in a data table (**TB,CFOAM,,,,PPR**) with the following input:

Constant	Meaning	Property
C1	ν^{pl}	Plastic Poisson's ratio

The valid range for the plastic Poisson's ratio is $-1 < \nu^{pl} < 0.5$.

Example 4.20: Defining the Crushable Foam Material Model

```
/prep7
mp, ex, 1,3e6
mp, nuxy, 1,0.2

p_c0 = 1e6
sigma_c0 = 5/4*p_c0
p_t0 = 4/5*p_c0
nu_p = 0.05
hardening_flag = 1
```

```

! yield stresses
TB,CFOAM,1,,,YIELD
TBDATA,1, sigma_c0, p_c0, p_t0

! plastic Poisson's ratio
TB,CFOAM,1,,,PPR
TBDATA,1, nu_p

! hardening type
TB,CFOAM,1,,,HTYPE
TBDATA,1, hardening_flag

! hardening curve
TB,CFOAM,1,1,,MHARD
TBPT,,0.0, 1.0
TBPT,,0.5, 2.2
TBPT,,1 , 2.9
TBPT,,1.5, 3.0
TBPT,,2 , 3.1

```

4.4.7.2. Resources

The following resource offers more information about crushable foam materials:

1. Deshpande, V. S. and N. A. Fleck. "Isotropic constitutive models for metallic foams." *Journal of the Mechanics and Physics of Solids*. 48.6-7 (2000): 1253-1283.

4.5. Rate-Dependent Plasticity (Viscoplasticity)

Use rate-dependent plasticity to model the rate-dependent plastic behavior of materials. Similar to [rate-independent plasticity](#) (p. 28), the material remains elastic when the stress is below the yield stress. Above the yield stress, the material behavior is similar to creep and higher plastic deformation rates are proportional to a higher equivalent stress in the material.

The following topics related to rate-dependent plasticity are available:

- 4.5.1. Perzyna and Peirce Options
- 4.5.2. Exponential Visco-Hardening (EVH) Option
- 4.5.3. Anand Option
- 4.5.4. Creep

For further information about rate-dependent plastic (viscoplastic) material options, see [Viscoplasticity in the Structural Analysis Guide](#).

4.5.1. Perzyna and Peirce Options

The Perzyna model has the following form:

$$\dot{\epsilon}_{pl} = \sum_{i=1}^n \gamma_i \left(\frac{\sigma}{\sigma_o} - 1 \right)^{1/m_i}$$

The Peirce model has the following form:

$$\dot{\epsilon}_{pl} = \sum_{i=1}^n \gamma_i \left(\left(\frac{\sigma}{\sigma_0} \right)^{1/m_i} - 1 \right)$$

where n is the number of terms, σ is the effective stress, $\dot{\epsilon}_{pl}$ is the equivalent plastic strain rate, m_i are the strain rate hardening parameters, γ_i are the material viscosity parameters, and σ_0 is the static yield stress.

As the material viscosity (γ_i) approaches ∞ , the strain rate hardening (m_i) approaches zero, or the equivalent plastic strain rate ($\dot{\epsilon}_{pl}$) approaches zero, the solution approaches the static (rate-independent) solution.

When the strain rate hardening is very small, the Peirce model has less difficulty converging as compared to the Perzyna model.

These models can be combined with an isotropic hardening model for hardening of the static yield stress (σ_0). Anisotropic viscoplastic behavior is simulated by combining the [Hill \(p. 38\)](#) yield surface.

For the Perzyna and Peirce constants, the number of terms (n) is determined from the number of material parameters defined in the table. Up to 20 terms are allowed and the material constants (defined via **TBDATA**) are:

Constant	Meaning	Property	Unit	Range
C1	m_1	Strain rate hardening exponent	Dimensionless	$0 < m_1 \leq 1$
C2	γ_1	Viscosity	1 / Time	$\gamma_1 > 0$
...	...	...	...	...
C(2n -1)	m_n	Strain rate hardening exponent	Dimensionless	$0 < m_n \leq 1$
C(2n)	γ_n	Viscosity	1 / Time	$\gamma_n > 0$

Specify the **PERZYNA** model as follows: **TB,RATE,,,,PERZYNA**

Specify the **PEIRCE** model as follows: **TB,RATE,,,,PEIRCE**

4.5.2. Exponential Visco-Hardening (EVH) Option

The EVH model has the following form:

$$\dot{\epsilon}_{pl} = \sum_{i=1}^n \left(\frac{\sigma - \bar{\sigma}_0}{K_i} \right)^{1/m_i}$$

where n is the number of terms, σ is the effective stress, $\dot{\epsilon}_{pl}$ is the equivalent plastic strain rate, m_i are the strain rate hardening parameters, K_i are the material viscosity parameters, and $\bar{\sigma}_0$ is the static yield stress:

$$\bar{\sigma}_0 = K_0 + R_0 \dot{\epsilon}_{pl} + R_\infty (1 - e^{-b \dot{\epsilon}_{pl}})$$

Where K_o is the initial yield stress, R_o is the linear hardening coefficient, R_∞ is the exponential hardening coefficient, and b is the saturation rate.

For the EVH constants, the number of terms (n) is determined from the number of material parameters defined in the table. Up to 20 terms are allowed and the material constants (defined via **TBDATA**) are:

Constant	Meaning	Property	Unit	Range
C1	K_o	Initial yield stress	Stress	$K_o > 0$
C2	R_o	Linear hardening coefficient	Stress	$R_o \geq 0$
C3	R_∞	Exponential hardening coefficient	Stress	--
C4	b	Exponential saturation rate	Dimensionless	$b > 0$
C5	m_1	Strain rate hardening exponent	Dimensionless	$0 < m_1 \leq 1$
C6	K_1	Viscosity	Stress $\times$ Time ^{m_1}	$K_1 > 0$
...	...	...	...	...
C(2n+3)	m_n	Strain rate hardening exponent	Dimensionless	$0 < m_n \leq 1$
C(2n+4)	K_n	Viscosity	Stress $\times$ Time ^{m_1}	$K_n > 0$

Specify the EVH model as follows: **TB,RATE,,,EVH**

4.5.3. Anand Option

The Anand option offers a unified plasticity model requiring no combination with other material models.

Details for the Anand option appear in [Anand Viscoplasticity Option](#) in the *Mechanical APDL Theory Reference*.

This option requires nine material constants input via the data-table command (**TBDATA**) in the order shown:

Constant	Meaning	Property	Units
C1	s_0	Initial value of deformation resistance	Stress
C2	Q/R	Q = Activation energy R = Universal gas constant	Energy / Volume Energy / Volume temperature
C3	A	Pre-exponential factor	1 / Time
C4	xi	Stress multiplier	Dimensionless
C5	m	Strain rate sensitivity of stress	Dimensionless
C6	h_0	Hardening / softening constant	Stress
C7	$\hat{S}$	Coefficient for deformation resistance saturation value	Stress

Constant	Meaning	Property	Units
C8	n	Strain rate sensitivity of saturation (deformation resistance) value	Dimensionless
C9	a	Strain rate sensitivity of hardening or softening	Dimensionless

Specify this option (*TBOPT* = ANAND) as follows: **TB**,RATE,,,9,ANAND

The Anand model supports plane strain, axisymmetric and full three-dimensional element behavior.

4.5.4. Creep

The creep strain rate, $\dot{\epsilon}_{cr}$, can be a function of stress, strain, temperature, and neutron flux level. Libraries of creep strain rate equations are included in [Implicit Creep Equations \(p. 82\)](#). Enter the constants shown in these equations using **TB**,CREEP and **TB**DATA as described below. These equations (expressed in incremental form) are characteristic of materials being used in creep design applications (see the [Mechanical APDL Theory Reference](#) for details).

For a list of the elements that support creep behavior, see [Material Model Element Support \(p. 5\)](#).

Three types of creep equations are available:

- Primary creep
- Secondary creep
- Irradiation induced creep

You can define the combined effects of more than one type of creep using the implicit equations specified via *TBOPT* = 11 or 12, or a user-defined creep equation.

Mechanical APDL analyzes creep via the implicit method. It is robust, fast, accurate, and recommended for general use, especially with problems involving large creep strain and large deformation. The method has provisions for including temperature-dependent constants. (Temperature dependency can also be incorporated via the Arrhenius option [**TB**FT,,,Option4].)

The program can model pure creep, creep with isotropic hardening plasticity, and creep with kinematic hardening plasticity, using both von Mises and Hill potentials. (See [Material Model Combinations \(p. 297\)](#).) Where the material model combination with creep is supported, any of the implicit creep models can be used.

For more information about this material option, see [Creep](#).

4.5.4.1. Implicit Creep Equations

Enter an implicit creep equation using **TBOPT** within the **TB** command. Enter the value of **TBOPT** corresponding to the equation, as shown in [Table 4.2: Implicit Creep Equations \(p. 82\)](#).

Table 4.2: Implicit Creep Equations

Creep Model (TB- OPT)	Name	Equation	Type
1	Strain Hardening	$\dot{\varepsilon}_{cr} = C_1 \sigma^{C_2} \varepsilon_{cr}^{C_3} e^{-C_4/T}$	$C_1 > 0$ Primary
2	Time Hardening	$\dot{\varepsilon}_{cr} = C_1 \sigma^{C_2} t^{C_3} e^{-C_4/T}$	$C_1 > 0$ Primary
3	Generalized Exponential	$\dot{\varepsilon}_{cr} = C_1 \sigma^{C_2} r e^{-rt}$, $r = C_5 \sigma^{C_3} e^{-C_4/T}$	$C_1 > 0$, $C_5 > 0$ Primary
4	Generalized Graham	$\dot{\varepsilon}_{cr} = C_1 \sigma^{C_2} (t^{C_3} + C_4 t^{C_5} + C_6 t^{C_7}) e^{-C_8/T}$	$C_1 > 0$ Primary
5	Generalized Blackburn	$\dot{\varepsilon}_{cr} = f(1 - e^{-rt}) + gt$ $f = C_1 e^{C_2 \sigma}$, $r = C_3 (\sigma / C_4)^{C_5}$, $g = C_6 e^{C_7 \sigma}$	$C_1 > 0$, $C_3 > 0$, $C_6 > 0$ Primary
6	Modified Time Hardening	$\varepsilon_{cr} = C_1 \sigma^{C_2} t^{C_3+1} e^{-C_4/T} / (C_3 + 1)$	$C_1 > 0$ Primary
7	Modified Strain Hardening	$\dot{\varepsilon}_{cr} = \{C_1 \sigma^{C_2} [(C_3 + 1) \varepsilon_{cr}]^{C_3}\}^{1/(C_3+1)} e^{-C_4/T}$	$C_1 > 0$ Primary
8	Generalized Garofalo	$\dot{\varepsilon}_{cr} = C_1 [\sinh(C_2 \sigma)]^{C_3} e^{-C_4/T}$	$C_1 > 0$ Secondary
9	Exponential form	$\dot{\varepsilon}_{cr} = C_1 e^{\sigma/C_2} e^{-C_3/T}$	$C_1 > 0$ Secondary
10	Norton	$\dot{\varepsilon}_{cr} = C_1 \sigma^{C_2} e^{-C_3/T}$	$C_1 > 0$ Secondary
11	Combined Time Hardening	$\varepsilon_{cr} = C_1 \sigma^{C_2} t^{C_3+1} e^{-C_4/T} / (C_3 + 1)$ $+ C_5 \sigma^{C_6} t e^{-C_7/T}$	$C_1 > 0$, $C_5 > 0$ Primary + Secondary
12	Rational polynomial	$\dot{\varepsilon}_{cr} = C_1 \frac{\partial \varepsilon_c}{\partial t}$, $\varepsilon_c = \frac{cpt}{1+pt} + \dot{\varepsilon}_m t$ $\dot{\varepsilon}_m = C_2 10^{C_3 \sigma} \sigma^{C_4}$	$C_2 > 0$ Primary + Secondary
13	Generalized Time Hardening	$c = C_7 \dot{\varepsilon}_m^{C_8} \sigma^{C_9}$, $p = C_{10} \dot{\varepsilon}_m^{C_{11}} \sigma^{C_{12}}$ $\varepsilon_{cr} = f t^r e^{-C_6/T}$ $f = C_1 \sigma + C_2 \sigma^2 + C_3 \sigma^3$	Primary
100	---	$r = C_4 + C_5 \sigma$ User Creep	---

where:

ε_{cr} = equivalent creep strain

$\dot{\varepsilon}_{cr}$ = change in equivalent creep strain with respect to time

σ = equivalent stress

T = temperature (absolute). The offset temperature (from **TOFFST**), is internally added to all temperatures for convenience.

C_1 through C_{12} = constants defined by the **TBDATA** command

t = time at end of substep

e = natural logarithm base

Specify the user-defined creep option by setting $TBOPT = 100$ and using **TB,STATE** to specify the number of state variables for the user-defined creep subroutine. See the [Guide to User-Programmable Features in the Programmer's Reference](#) for more information. The **RATE** command is necessary to activate implicit creep for specific elements (see the **RATE** command description for details).

For temperature-dependent constants, define the temperature using **TBTEMP** for each set of data. Then, define constants C_1 through C_m using **TBDATA** (where m is the number of constants, and depends on the creep model you choose).

The following example shows how you would define the implicit creep model represented by $TBOPT = 1$ at two temperature points.

```

TB,CREEP,1,,1      !Activate creep data table, specify creep model 1
TBTEMP,100         !Define first temperature
TBDATA,1,c11,c12,c13,c14 !Creep constants c11, c12, c13, c14 at first temp.
TBTEMP,200         !Define second temperature
TBDATA,1,c21,c22,c23,c24 !Creep constants c21, c22, c23, c24 at second temp.

```

Coefficients are linearly interpolated for temperatures that fall between user defined **TBTEMP** values. For some creep models, where the change in coefficients spans several orders of magnitude, this linear interpolation might introduce inaccuracies in solution results. Use enough curves to accurately capture the temperature dependency. Also, consider using the curve fitting subroutine to calculate a temperature dependent coefficient that includes the Arrhenius term.

When a temperature is above or below the range of defined temperature values, the coefficients for the nearest temperature are used with no extrapolation.

For a list of elements that can be used with this material option, see [Material Model Element Support \(p. 5\)](#).

See [Creep](#) in the *Structural Analysis Guide* for more information on this material option.

4.6. Hyperelasticity

Hyperelastic material behavior is supported by [current-technology elements](#) with a three-dimensional strain state, including 3-D solid elements and plane strain, axisymmetric, elbow and thick pipe elements. For a list of elements that can be used with hyperelastic material models, see [Material Model Element Support \(p. 5\)](#). You can specify options to describe the hyperelastic material behavior for these elements.

Hyperelasticity options are available via the $TBOPT$ argument on the **TB,HYPER** command.

Several forms of strain energy potentials describe the hyperelasticity of materials. These are based on either strain invariants or principal stretches. The behavior of materials is assumed to be incompressible or nearly incompressible.

The following hyperelastic material model topics are available:

- 4.6.1. Arruda-Boyce Hyperelasticity
- 4.6.2. Blatz-Ko Foam Hyperelasticity
- 4.6.3. Extended Tube Hyperelasticity
- 4.6.4. Gent Hyperelasticity
- 4.6.5. Mooney-Rivlin Hyperelasticity
- 4.6.6. Neo-Hookean Hyperelasticity
- 4.6.7. Ogden Hyperelasticity
- 4.6.8. Ogden Compressible Foam Hyperelasticity
- 4.6.9. Polynomial Form Hyperelasticity
- 4.6.10. Response Function Hyperelasticity
- 4.6.11. Yeoh Hyperelasticity
- 4.6.12. Special Hyperelasticity

For information about other hyperelastic material models, see [Special Hyperelasticity \(p. 93\)](#).

4.6.1. Arruda-Boyce Hyperelasticity

The **TB,HYPER,,,,BOYCE** option uses the Arruda-Boyce form of strain energy potential given by:

$$W = \mu \left[\frac{1}{2} (\bar{I}_1 - 3) + \frac{1}{20\lambda_L^2} (\bar{I}_1^2 - 9) + \frac{11}{1050\lambda_L^4} (\bar{I}_1^3 - 27) + \frac{19}{7000\lambda_L^6} (\bar{I}_1^4 - 81) + \frac{519}{673750\lambda_L^8} (\bar{I}_1^5 - 243) \right] + \frac{1}{d} \left(\frac{J^2 - 1}{2} - \ln J \right)$$

where:

- W = strain energy per unit reference volume
- $\bar{I}_1$ = first deviatoric strain invariant
- J = determinant of the elastic deformation gradient $\mathbf{F}$
- μ = initial shear modulus of materials
- λ_L = limiting network stretch
- d = material incompressibility parameter

The initial bulk modulus is defined as:

$$K = \frac{2}{d}$$

As λ_L approaches infinity, the option becomes equivalent to the Neo-Hookean option.

The constants μ , λ_L , and d are defined by C1, C2, and C3 (**TBDATA**).

For a list of elements that can be used with this material option, see [Material Model Element Support \(p. 5\)](#).

For more information about this material option, see [Arruda-Boyce Hyperelastic Option](#).

4.6.2. Blatz-Ko Foam Hyperelasticity

The **TB,HYPER,,,,BLATZ** option uses the Blatz-Ko form of strain energy potential given by:

$$W = \frac{\mu}{2} \left(\frac{I_2}{I_3} + 2\sqrt{I_3} - 5 \right)$$

where:

W = strain energy per unit reference volume

μ = initial strain shear modulus

I_2 and I_3 = second and third strain invariants

The initial bulk modulus k is defined as:

$$k = \frac{5}{3}\mu$$

The model has only one constant μ and is defined by C1 using the **TB,DATA** command.

See [Blatz-Ko Foam Hyperelastic Option](#) in the *Structural Analysis Guide* for more information on this material option.

4.6.3. Extended Tube Hyperelasticity

The extended tube model is available as a hyperelastic material option (**TB,HYPER**). The model simulates filler-reinforced elastomers and other rubber-like materials, supports material curve-fitting, and is available in all [current-technology elements](#) continuum, shell, and pipe elements.

Five material constants are needed for the extended-tube model:

TBOPT	Constants	Purpose
C1	G_c	Crosslinked network modulus
C2	G_e	Constraint network modulus
C3	β	Empirical parameter ($0 \leq \beta \leq 1$)
C4	δ	Extensibility parameter
C5	d	Incompressibility parameter

Following the material data-table command (**TB**), specify the material constant values via the **TB,DATA** command, as shown in this example:

```
TB,HYPER,1,,5,ETUBE      ! Hyperelastic material, 1 temperature,
                          ! 5 material constants, and the extended tube option
TBDATA,1,0.25, 0.8,1.0,0.5,1.0e-5  ! Five material constant values (C1 through C5)
```

For more information, see the documentation for the **TB,HYPER** command, and [Extended Tube Model](#) in the *Mechanical APDL Theory Reference*.

4.6.4. Gent Hyperelasticity

The **TB,HYPER,,,GENT** option uses the Gent form of strain energy potential given by:

$$W = -\frac{\mu J_m}{2} \ln \left(1 - \frac{\bar{I}_1 - 3}{J_m} \right) + \frac{1}{d} \left(\frac{J^2 - 1}{2} - \ln J \right)$$

where:

W = strain energy per unit reference volume

μ = initial shear modulus of material

J_m = limiting value of $\bar{I}_1 - 3$

$\bar{I}_1$ = first deviatoric strain invariant

J = determinant of the elastic deformation gradient $\mathbf{F}$

d = material incompressibility parameter

The initial bulk modulus K is defined as:

$$K = \frac{2}{d}$$

As J_m approaches infinity, the option becomes equivalent to the Neo-Hookean option.

The constants μ , J_m , and d are defined by C1, C2, and C3 using the **TBDATA** command.

For a list of elements that can be used with this material option, see [Material Model Element Support](#) (p. 5).

See [Gent Hyperelastic Option](#) in the *Structural Analysis Guide* for more information on this material option.

4.6.5. Mooney-Rivlin Hyperelasticity

The Mooney-Rivlin model applies to [current-technology](#) shell, beam, solid, and plane elements.

The **TB,HYPER,,,MOONEY** option allows you to define 2, 3, 5, or 9 parameter Mooney-Rivlin models using $NPTS = 2, 3, 5, \text{ or } 9$, respectively.

For $NPTS = 2$ (2 parameter Mooney-Rivlin option, which is also the default), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + \frac{1}{d}(J - 1)^2$$

where:

W = strain energy potential

$\bar{I}_1$ = first deviatoric strain invariant

$\bar{I}_2$ = second deviatoric strain invariant

c_{10}, c_{01} = material constants characterizing the deviatoric deformation of the material

d = material incompressibility parameter

The initial shear modulus is defined as:

$$\mu = 2(c_{10} + c_{01})$$

and the initial bulk modulus is defined as:

$$K = \frac{2}{d}$$

where:

$$d = (1 - 2\nu) / (C_{10} + C_{01})$$

The constants c_{10} , c_{01} , and d are defined by C1, C2, and C3 (**TBDATA**).

For $NPTS = 3$ (3 parameter Mooney-Rivlin option, which is also the default), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + c_{11}(\bar{I}_1 - 3)(\bar{I}_2 - 3) + \frac{1}{d}(J - 1)^2$$

The constants c_{10} , c_{01} , c_{11} ; and d are defined by C1, C2, C3, and C4 (**TBDATA**).

For $NPTS = 5$ (5 parameter Mooney-Rivlin option), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + c_{20}(\bar{I}_1 - 3)^2 + c_{11}(\bar{I}_1 - 3)(\bar{I}_2 - 3) + c_{02}(\bar{I}_2 - 3)^2 + \frac{1}{d}(J - 1)^2$$

The constants c_{10} , c_{01} , c_{20} , c_{11} , c_{02} , and d are material constants defined by C1, C2, C3, C4, C5, and C6 (**TBDATA**).

For $NPTS = 9$ (9 parameter Mooney-Rivlin option), the form of the strain energy potential is:

$$W = c_{10}(\bar{I}_1 - 3) + c_{01}(\bar{I}_2 - 3) + c_{20}(\bar{I}_1 - 3)^2 + c_{11}(\bar{I}_1 - 3)(\bar{I}_2 - 3) + c_{02}(\bar{I}_2 - 3)^2 + c_{30}(\bar{I}_1 - 3)^3 + c_{21}(\bar{I}_1 - 3)^2(\bar{I}_2 - 3) + c_{12}(\bar{I}_1 - 3)(\bar{I}_2 - 3)^2 + c_{03}(\bar{I}_2 - 3)^3 + \frac{1}{d}(J - 1)^2$$

The constants c_{10} , c_{01} , c_{20} , c_{11} , c_{02} , c_{30} , c_{21} , c_{12} , c_{03} , and d are material constants defined by C1, C2, C3, C4, C5, C6, C7, C8, C9, and C10 (**TBDATA**).

See [Mooney-Rivlin Hyperelastic Option \(TB, HYPER\)](#) in the *Structural Analysis Guide* for more information.

4.6.6. Neo-Hookean Hyperelasticity

The option **TB,HYPER,,,,NEO** uses the Neo-Hookean form of strain energy potential, which is given by:

$$W = \frac{\mu}{2} (\bar{I}_1 - 3) + \frac{1}{d} (J - 1)^2$$

where:

W = strain energy per unit reference volume

$\bar{I}_1$ = first deviatoric strain invariant

μ = initial shear modulus of the material

d = material incompressibility parameter.

J = determinant of the elastic deformation gradient **F**

The initial bulk modulus is defined by:

$$K = \frac{2}{d}$$

The constants μ and d are defined via the **TBDATA** command.

See [Neo-Hookean Hyperelastic Option](#) in the *Structural Analysis Guide* for more information on this material option.

4.6.7. Ogden Hyperelasticity

The **TB,HYPER,,,,OGDEN** option uses the Ogden form of strain energy potential. The Ogden form is based on the principal stretches of the left Cauchy-Green tensor. The strain energy potential is:

$$W = \sum_{i=1}^N \frac{\mu_i}{\alpha_i} (\bar{\lambda}_1^{\alpha_i} + \bar{\lambda}_2^{\alpha_i} + \bar{\lambda}_3^{\alpha_i} - 3) + \sum_{k=1}^N \frac{1}{d_k} (J - 1)^{2k}$$

where:

W = strain energy potential

$\bar{\lambda}_p$ ($p = 1, 2, 3$) = deviatoric principal stretches, defined as $\bar{\lambda}_p = J^{\frac{1}{3}} \lambda_p$

λ_p = principal stretches of the left Cauchy-Green tensor

J = determinant of the elastic deformation gradient

N, μ_p, α_p and d_p = material constants

In general there is no limitation on the value of N . (See the **TB** command.) A higher value of N can provide a better fit to the exact solution. It may however cause numerical difficulties in fitting the material constants. For this reason, very high values of N are not recommended.

The initial shear modulus μ is defined by:

$$\mu = \frac{1}{2} \sum_{i=1}^N \alpha_i \mu_i$$

The initial bulk modulus K is defined by:

$$K = \frac{2}{d_1}$$

For N = 1 and $\alpha_1 = 2$, the Ogden option is equivalent to the Neo-Hookean option. For N = 2, $\alpha_1 = 2$, and $\alpha_2 = -2$, the Ogden option is equivalent to the 2 parameter Mooney-Rivlin option.

The constants μ_p , α_p and d_p are defined using the **TBDATA** command in the following order:

For N (NPTS) = 1:

$$\mu_1, \alpha_1, d_1$$

For N (NPTS) = 2:

$$\mu_1, \alpha_1, \mu_2, \alpha_2, d_1, d_2$$

For N (NPTS) = 3:

$$\mu_1, \alpha_1, \mu_2, \alpha_2, \mu_3, \alpha_3, d_1, d_2, d_3$$

For N (NPTS) = k:

$$\mu_1, \alpha_1, \mu_2, \alpha_2, \dots, \mu_k, \alpha_k, d_1, d_2, \dots, d_k$$

See [Ogden Hyperelastic Option](#) in the *Structural Analysis Guide* for more information on this material option.

4.6.8. Ogden Compressible Foam Hyperelasticity

The **TB,HYPER,,,,FOAM** option uses the Ogden form of strain energy potential for highly compressible elastomeric foam material. The strain energy potential is based on the principal stretches of the left Cauchy-Green tensor and is given by:

$$W = \sum_{i=1}^N \frac{\mu_i}{\alpha_i} (J^{\alpha_i/3} (\bar{\lambda}_1^{\alpha_i} + \bar{\lambda}_2^{\alpha_i} + \bar{\lambda}_3^{\alpha_i}) - 3) + \sum_{i=1}^N \frac{\mu_i}{\alpha_i \beta_i} (J^{-\alpha_i \beta_i} - 1)$$

where:

W = strain energy potential

$\bar{\lambda}_p^{\alpha_i}$ (p=1,2,3)=deviatoric principal stretch

J = determinant of the elastic deformation gradient

N, μ_i , α_i and β_i = material constants

For this material option, the volumetric and deviatoric terms are tightly coupled. Hence, this model is meant to simulate highly compressible elastomers.

In general there is no limitation on the value of N. (See the **TB** command.) A higher value of N can provide a better fit to the exact solution. It may however cause numerical difficulties in fitting the material constants. For this reason, very high values of N are not recommended.

The initial shear modulus μ is defined by:

$$\mu = \frac{\sum_{i=1}^N \mu_i \alpha_i}{2}$$

and the initial bulk modulus K is defined by:

$$K = \sum_{i=1}^N \mu_i \alpha_i \left(\frac{1}{3} + \beta_i \right)$$

For N = 1, $\alpha_1 = -2$, $\mu_1 = -\mu$, and $\beta_1 = 0.5$, the Ogden foam option is equivalent to the Blatz-Ko option.

The constants μ_i , α_i and β_i are defined using the **TBDATA** command in the following order:

For N (*NPTS*) = 1:

$$\mu_1, \alpha_1, \beta_1$$

For N (*NPTS*) = 2:

$$\mu_1, \alpha_1, \mu_2, \alpha_2, \beta_1, \beta_2$$

For N (*NPTS*) = 3:

$$\mu_1, \alpha_1, \mu_2, \alpha_2, \mu_3, \alpha_3, \beta_1, \beta_2, \beta_3$$

For N (*NPTS*) = k:

$$\mu_1, \alpha_1, \mu_2, \alpha_2, \dots, \mu_k, \alpha_k, \beta_1, \beta_2, \dots, \beta_k$$

See [Ogden Compressible Foam Hyperelastic Option](#) in the *Structural Analysis Guide* for more information on this material option.

4.6.9. Polynomial Form Hyperelasticity

The **TB,HYPER,,,POLY** option allows you to define a polynomial form of strain energy potential. The form of the strain energy potential for the Polynomial option is given by:

$$W = \sum_{i+j=1}^N c_{ij} (\bar{I}_1 - 3)^i (\bar{I}_2 - 3)^j + \sum_{k=1}^N \frac{1}{d_k} (J - 1)^{2k}$$

where:

W = strain energy potential

$\bar{I}_1$ = first deviatoric strain invariant

$\bar{I}_2$ = second deviatoric strain invariant

J = determinant of the elastic deformation gradient **F**

N, c_{ij} , and d = material constants

In general there is no limitation on the value of N. (See the **TB** command.) A higher value of N can provide a better fit to the exact solution. It may however cause a numerical difficulty in fitting the material constants, and it also requests enough data to cover the whole range of deformation for which you may be interested. For these reasons, a very high value of N is not recommended.

The initial shear modulus μ is defined by:

$$\mu = 2(c_{10} + c_{01})$$

and the initial bulk modulus is defined as:

$$K = \frac{2}{d_1}$$

For N = 1 and $c_{01} = 0$, the polynomial form option is equivalent to the Neo-Hookean option. For N = 1, it is equivalent to the 2 parameter Mooney-Rivlin option. For N = 2, it is equivalent to the 5 parameter Mooney-Rivlin option, and for N = 3, it is equivalent to the 9 parameter Mooney-Rivlin option.

The constants c_{ij} and d are defined using the **TBDATA** command in the following order:

For N (NPTS) = 1:

$$c_{10}, c_{01}, d_1$$

For N (NPTS) = 2:

$$c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, d_1, d_2$$

For N (NPTS) = 3:

$$c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, c_{30}, c_{21}, c_{12}, c_{03}, d_1, d_2, d_3$$

For N (NPTS) = k:

$$c_{10}, c_{01}, c_{20}, c_{11}, c_{02}, c_{30}, c_{21}, c_{12}, c_{03}, \dots, c_{k0}, c_{(k-1)1}, \dots, c_{0k}, d_1, d_2, \dots, d_k$$

See [Polynomial Form Hyperelastic Option](#) in the *Structural Analysis Guide* for more information on this material option.

4.6.10. Response Function Hyperelasticity

The response function option for hyperelastic material constants (**TB,HYPER,,,,RESPONSE**) uses experimental data (**TB,EXPE**) to determine the constitutive response functions.

The response functions (first derivatives of the hyperelastic potential) are used to determine hyperelastic constitutive behavior of the material. In general, the stiffness matrix requires derivatives of the response functions (second derivatives of the hyperelastic potential).

The method for determining the derivatives is ill-conditioned near the zero stress-strain point; therefore, a deformation limit is used, below which the stiffness matrix is calculated with only the response functions. The deformation measure is $\delta = I_1 - 3$, where I_1 is the first invariant of the Cauchy-Green deformation tensor.

The stiffness matrix is then calculated with only the response functions if $\delta < C_1$, where C_1 is the material constant deformation limit (default 1×10^{-5}).

The remaining material parameters are for the volumetric strain energy potential, given by

$$W(J) = \sum_{k=1}^N \frac{1}{d_k} (J-1)^{2k}$$

where N is the *NPTS* value (**TB,HYPER,,,,RESPONSE**) and d_k represents the material constants incompressibility parameters (default 0.0) and J is the volume ratio. Use of experimental volumetric data requires *NPTS* = 0. Incompressible behavior results if all $d_k = 0$ or *NPTS* = 0 with no experimental volumetric data.

4.6.11. Yeoh Hyperelasticity

The **TB,HYPER,,,,YEOH** option follows a reduced polynomial form of strain energy potential by Yeoh. The form of the strain energy potential for the Yeoh option is given by:

$$W = \sum_{i=1}^N c_{i0} (\bar{I}_1 - 3)^i + \sum_{k=1}^N \frac{1}{d_k} (J-1)^{2k}$$

where:

W = strain energy potential

$\bar{I}_1$ = first deviatoric strain invariant

J = determinant of the elastic deformation gradient **F**

N , c_{i0} , and d_k = material constants

In general there is no limitation on the value of N . (See the **TB** command.) A higher value of N can provide a better fit to the exact solution. It may however cause a numerical difficulty in fitting the material constants, and it also requests enough data to cover the whole range of deformation for which you may be interested. For these reasons, a very high value of N is not recommended.

The initial shear modulus μ is defined by:

$$\mu = 2c_{10}$$

and the initial bulk modulus K is defined as:

$$K = \frac{2}{d_1}$$

For $N = 1$ the Yeoh form option is equivalent to the Neo-Hookean option.

The constants c_{i0} and d_k are defined using the **TB**DATA command in the following order:

For N (NPTS) = 1:

$$c_{10}, d_1$$

For N (NPTS) = 2:

$$c_{10}, c_{20}, d_1, d_2$$

For N (NPTS) = 3:

$$c_{10}, c_{20}, c_{30}, d_1, d_2, d_3$$

For N (NPTS) = k:

$$c_{10}, c_{20}, c_{30}, \dots, c_{k0}, d_1, d_2, \dots, d_k$$

See [Yeoh Hyperelastic Option](#) in the *Structural Analysis Guide* for more information on this material option.

4.6.12. Special Hyperelasticity

The following hyperelastic material models have their own *Lab* value on the **TB** command (and are not simply **TBOPT** hyperelasticity options on the **TB,HYPER** command):

[4.6.12.1. Anisotropic Hyperelasticity](#)

[4.6.12.2. Bergstrom-Boyce Material](#)

[4.6.12.3. Mullins Effect](#)

[4.6.12.4. User-Defined Hyperelastic Material](#)

4.6.12.1. Anisotropic Hyperelasticity

The anisotropic hyperelasticity material model (**TB,AHYPER**) is available with [current-technology shell, plane, and solid elements](#) (p. 5). Anisotropic hyperelasticity is a potential-based-function with parameters to define the volumetric part, the isochoric part and the material directions.

Two strain energy potentials, as forms of polynomial or exponential function, are available for characterizing the isochoric part of strain energy potential.

You can use anisotropic hyperelasticity to model elastomers with reinforcements, and for biomedical materials such as muscles or arteries.

The strain energy potential for anisotropic hyperelasticity is given by:

$$W = W_v(J) + W_d(\bar{C}, A \otimes A, B \otimes B)$$

The volumetric strain energy is given by:

$$W_v(J) = \frac{1}{d} \cdot (J-1)^2$$

The polynomial-function-based strain energy potential is given by:

$$W_d(C, A \otimes A, B \otimes B) = \sum_{i=1}^3 a_i (\bar{I}_1 - 3)^i + \sum_{j=1}^3 b_j (\bar{I}_2 - 3)^j + \sum_{k=2}^6 c_k (\bar{I}_4 - 1)^k \\ + \sum_{l=2}^6 d_l (\bar{I}_5 - 1)^l + \sum_{m=2}^6 e_m (\bar{I}_6 - 1)^m + \sum_{n=2}^6 f_n (\bar{I}_7 - 1)^n + \sum_{o=2}^6 g_o (\bar{I}_8 - \zeta)^o$$

The exponential-function-based strain energy potential is given by:

$$W_d(\bar{C}, A \otimes A, B \otimes B) = \sum_{i=1}^3 a_i (\bar{I}_1 - 3)^i + \sum_{j=1}^3 b_j (\bar{I}_2 - 3)^j + \frac{c_1}{2c_2} \{ \exp[c_2 (\bar{I}_4 - 1)^2] - 1 \} \\ + \frac{e_1}{2e_2} \{ \exp[e_2 (\bar{I}_6 - 1)^2] - 1 \}$$

Use **TB, AHYPER,, TBOPT** to define the isochoric part, material directions and the volumetric part. Only one TB table can be defined for each option. You can either define polynomial or exponential strain energy potential.

TBOPT	Constants	Purpose	Input Format
POLY	C1 to C31	Anisotropic strain energy potential	TB, AHYPER, , , , POLY TBDATA, , , A1, A2, A3, B1 . . .
EXP	C1 to C10	Exponential anisotropic strain energy potential	TB, AHYPER, , , , EXP0 TBDATA, 1, A1, A2, A3, B1, B2, B3 TBDATA, 7, C1, C2, E1, E2
AVEC	C1 to C3	Material direction constants	TB, AHYPER, , , , AVEC TBDATA, , , A1, A2, A3
BVEC	C1 to C3	Material direction constants	TB, AHYPER, , , , BVEC TBDATA, , , B1, B2, B3
PVOL	C1	Volumetric potential	TB, AHYPER, , , , PVOL TBDATA, , , D

You can enter temperature-dependent data for anisotropic hyperelastic material via the **TBTEMP** command. For the first temperature curve, issue **TB, AHYPER,,, TBOPT**, then input the first temperature (**TBTEMP**). The subsequent **TBDATA** command inputs the data.

The program interpolates the temperature data to the material points automatically using linear interpolation. When the temperature is out of the specified range, the closest temperature point is used.

For more information, see the **TB** command, and [Anisotropic Hyperelasticity](#) in the *Mechanical APDL Theory Reference*.

4.6.12.2. Bergstrom-Boyce Material

The Bergstrom Boyce option (**TB, BB**) is a phenomenological-based, highly nonlinear, rate-dependent material model for simulation of elastomer materials. The model assumes inelastic response only for shear distortional behavior defined by an isochoric strain energy potential, while the response to volumetric deformations is still purely elastic and characterized by a volumetric strain energy potential.

This model requires seven material constants input for the isochoric ($TBOPT = ISO$) option and one material constant for the volumetric potential ($TBOPT = PVOL$) option. Issue the **TBDATA** data-table command to input the constant values in the order shown:

Isochoric

TB,BB,,,,ISO

Constant	Meaning	Property	Units
C1	μ_0	Initial shear modulus for Part A	Pa
C2	N_0	$(\lambda_A^{\text{lock}})^2$, where λ^{lock} is the limiting chain stretch	Dimensionless
C3	μ_1	Initial shear modulus for Part B	Pa
C4	N_1	$(\lambda_B^{\text{lock}})^2$	Dimensionless
C5	$\dot{\gamma}_0 / \tau_{\text{base}}^m$	Material constant	$\text{s}^{-1}(\text{Pa})^{-m}$
C6	c	Material constant	Dimensionless
C7	m	Material constant	Dimensionless
C8	ε	Optional material constant	Dimensionless

The default optional material constant is $\varepsilon = 1 \times 10^{-5}$. However, if $TBNPT > 7$ or $TBNPT$ is unspecified, the table value is used instead. If the table value is zero or exceeds 1×10^{-3} , the default constant value is used.

Volumetric Potential

TB,BB,,,,PVOL

Constant	Meaning	Property	Units
C1	d	$1 / K$, where K is the bulk modulus	$1 / \text{Pa}$

For more information, see:

- The BB argument and associated specifications in the **TB** command documentation
- [Bergstrom-Boyce Hyperviscoelastic Material Model](#) in the *Structural Analysis Guide*
- [Bergstrom-Boyce](#) in the *Mechanical APDL Theory Reference*

4.6.12.3. Mullins Effect

The Mullins effect is a modification to the nearly- and fully-incompressible isotropic [hyperelastic](#) (p. 83) constitutive models (all **TB,HYPER** options with the exception of $TBOPT = \text{BLATZ}$ or $TBOPT = \text{FOAM}$) and is used with those models. The data table is initiated via the following command:

TB,CDM,MAT,NTEMPS,NPTS,TBOPT

The material constants for each valid $TBOPT$ value follow:

Modified Ogden-Roxburgh Pseudo-Elastic		
TBOPT = PSE2		
Constant	Meaning	Property
C1	r	Damage variable parameter
C2	m	Damage variable parameter
C3	β	Damage variable parameter

For more information, see:

- The CDM argument and associated specifications in the **TB** command documentation
- [Mullins Effect Material Model](#) in the *Structural Analysis Guide*
- [Mullins Effect](#) in the *Mechanical APDL Theory Reference*.

4.6.12.4. User-Defined Hyperelastic Material

User-defined isotropic and anisotropic strain-energy potentials are specified via **TBOPT = USER** on the **TB,HYPER** or **TB,AHYPER** commands, enabling you to define a strain-energy potential and then program the subroutine [UserHyper](#) or [UserHyperAniso](#) to define the derivatives of the strain-energy potential with respect to the strain invariants.

For more information about writing a user-defined hyperelasticity subroutine, see [Subroutine UserHyper \(Writing Your Own Isotropic Hyperelasticity Laws\)](#) and [Subroutine UserHyperAniso \(Writing Your Own Anisotropic Hyperelasticity Laws\)](#) in the [Guide to User-Programmable Features in the Programmer's Reference](#).

4.7. Viscoelasticity

Viscoelastic materials are characterized by a combination of elastic behavior, which stores energy during deformation, and viscous behavior, which dissipates energy during deformation.

The elastic behavior is rate-independent and represents the recoverable deformation due to mechanical loading. The viscous behavior is rate-dependent and represents dissipative mechanisms within the material.

A wide range of materials (such as polymers, glassy materials, soils, biologic tissue, and textiles) exhibit viscoelastic behavior.

Following are descriptions of the viscoelastic constitutive models, which include both small- and large-deformation formulations. Also presented is time-temperature superposition for thermorheologically simple materials and a harmonic domain viscoelastic model.

4.7.1. Viscoelastic Formulation

4.7.2. Time-Temperature Superposition

4.7.3. Harmonic Viscoelasticity

For more information, see [Viscoelasticity](#) in the *Structural Analysis Guide*.

4.7.1. Viscoelastic Formulation

The following formulation topics for viscoelasticity are available:

[4.7.1.1. Small Deformation](#)

[4.7.1.2. Small Strain with Large Deformation](#)

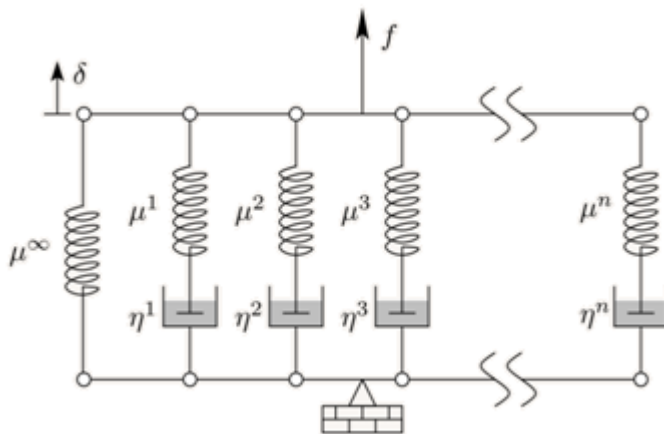
[4.7.1.3. Large Deformation](#)

[4.7.1.4. Stored Energy and Dissipation](#)

4.7.1.1. Small Deformation

The following figure shows a one dimensional representation of a generalized Maxwell solid. It consists of a spring element in parallel with a number of spring and dashpot Maxwell elements.

Figure 4.22: Generalized Maxwell Solid in One Dimension



The spring stiffnesses are μ^i , the dashpot viscosities are η^i , and the relaxation time is defined as the ratio of viscosity to stiffness, $\tau^i = \eta^i / \mu^i$.

In three dimensions, the constitutive model for a generalized Maxwell model is given by:

$$\sigma = \int_0^t 2G(t-\tau) \frac{d\mathbf{e}}{d\tau} d\tau + \mathbf{I} \int_0^t K(t-\tau) \frac{d\Delta}{d\tau} d\tau \quad (4.12)$$

where:

σ = Cauchy stress

$\mathbf{e}$ = deviatoric strain

Δ = volumetric strain

τ = past time

$\mathbf{I}$ = identity tensor

and $G(t)$ and $K(t)$ are the Prony series shear and bulk-relaxation moduli, respectively:

$$G(t) = G_0 \left[\alpha_\infty^G + \sum_{i=1}^{n_G} \alpha_i^G \exp\left(-\frac{t}{\tau_i^G}\right) \right] \quad (4.13)$$

$$K(t) = K_0 \left[\alpha_\infty^K + \sum_{i=1}^{n_K} \alpha_i^K \exp\left(-\frac{t}{\tau_i^K}\right) \right] \quad (4.14)$$

where:

G_0, K_0 = relaxation moduli at $t = 0$

n_G, n_K = number of Prony terms

α_i^G, α_i^K = relative moduli

τ_i^G, τ_i^K = relaxation time

For use in the incremental finite element procedure, the solution for Equation 4.12 (p. 97) at $t_1 = t_0 + \Delta t$ is:

$$s_i(t_1) = s_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^G}\right) + \int_{t_0}^{t_1} G_0 \alpha_i^G \exp\left(-\frac{t_1 - \tau}{\tau_i^G}\right) \frac{de}{d\tau} d\tau \quad (4.15)$$

$$p_i(t_1) = p_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^K}\right) + \int_{t_0}^{t_1} K_0 \alpha_i^K \exp\left(-\frac{t_1 - \tau}{\tau_i^K}\right) \frac{d\Delta}{d\tau} d\tau \quad (4.16)$$

where s_i and p_i are the deviatoric and pressure components, respectively, of the Cauchy stress for each Maxwell element.

By default, the midpoint rule is used to approximate the integrals:

$$s_i(t_1) = s_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^G}\right) + 2G_0 \alpha_i^G \exp\left(-\frac{\Delta t}{2\tau_i^G}\right) (e(t_1) - e(t_0)) \quad (4.17)$$

$$p_i(t_1) = p_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^K}\right) + 2K_0 \alpha_i^K \exp\left(-\frac{\Delta t}{2\tau_i^K}\right) (\Delta(t_1) - \Delta(t_0)) \quad (4.18)$$

An alternative stress integration method is to assume a constant strain rate over the time increment. Then the stress update is:

$$s_i(t_1) = s_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^G}\right) + 2G_0 \alpha_i^G \frac{\tau_i^G}{\Delta t} \left(1 - \exp\left(-\frac{\Delta t}{\tau_i^G}\right)\right) (e(t_1) - e(t_0)) \quad (4.19)$$

$$p_i(t_1) = p_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^K}\right) + 2K_0 \alpha_i^K \frac{\tau_i^K}{\Delta t} \left(1 - \exp\left(-\frac{\Delta t}{\tau_i^K}\right)\right) (\Delta(t_1) - \Delta(t_0)) \quad (4.20)$$

The model requires input of the parameters in Equation 4.13 (p. 98) and Equation 4.14 (p. 98). The relaxation moduli at $t = 0$ are obtained from the elasticity parameters input using the **MP** command or via an elastic data table (**TB,ELASTIC**). The Prony series relative moduli and relaxation times are input via a Prony data table (**TB,PRONY**), and separate data tables are necessary for specifying the bulk and shear Prony parameters.

Temperature-dependent elastic properties can be defined, but Equation 4.17 (p. 98) through Equation 4.20 (p. 98) assume that they are constant. The error in the approximations can be significant if the rate of change for the elastic properties is significant compared to the relaxation times.

For the shear Prony data table, $TBOPT = \text{SHEAR}$, $NPTS = n_G$, and the constants in the data table follow this pattern:

Table Location	Constant
1	α_1^G
2	τ_1^G
...	...
$2(NPTS) - 1$	α_{nG}^G
$2(NPTS)$	τ_{nG}^G

For the bulk Prony data table, $TBOPT = \text{BULK}$, $NPTS = n_K$, and the constants in the data table follow this pattern:

Table Location	Constant
1	α_1^K
2	τ_1^K
...	...
$2(NPTS) - 1$	α_{nK}^K
$2(NPTS)$	τ_{nK}^K

Use $TBOPT = \text{INTEGRATION}$ with the Prony table (**TB**,PRONY) to select the stress update algorithm. If the table is not defined, or the value of the first table location is equal to 1, then the default midpoint formula from Equation 4.17 (p. 98) and Equation 4.18 (p. 98) are used. If the value of the first table location is equal to 2, then the constant strain rate formula from Equation 4.19 (p. 98) and Equation 4.20 (p. 98) are used.

4.7.1.2. Small Strain with Large Deformation

This model is used when the large-deflection effects are active (**NLGEOM**,ON).

To account for large displacement, the model is formulated in the co-rotated configuration using the co-rotated deviatoric stress $\Sigma = R^T s R$, where R is the rotation obtained from the polar decomposition of the deformation gradient. The pressure component of the Cauchy stress does not need to account for the material rotation and uses the same formulation as the small-deformation model.

The deviatoric stress update is then expressed as:

$$s_i = \Delta R \Sigma_i(t_0) \Delta R^T \exp\left(-\frac{\Delta t}{\tau_i^G}\right) + 2G_0 \alpha_i^G \exp\left(-\frac{\Delta t}{2\tau_i^G}\right) (e(t_1) - \Delta R e(t_0) \Delta R^T) \quad (4.21)$$

where $\Delta R = R(t_1)R^T(t_0)$ is the incremental rotation.

Parameter input for this model resembles the input requirements for the [small-deformation viscoelastic \(p. 97\)](#) model.

4.7.1.3. Large Deformation

The large-strain viscoelastic constitutive model is a modification of the model proposed by [Simo](#). Modifications are included for viscoelastic volumetric response and the use of [time-temperature superposition](#) (p. 102).

The linear structure of the formulation is provided by the generalized Maxwell model. Extension to large deformation requires only a hyperelastic model for the springs in the Maxwell elements. Hyperelasticity is defined by a strain energy potential where, for isotropic materials:

$$W(C) = w(\bar{C}) + U(J) \quad (4.22)$$

where:

C = right Cauchy-Green deformation tensor

$\bar{C}$ = isochoric part of C

J = determinant of the deformation gradient

The second Piola-Kirchhoff stress in the Maxwell element springs is then:

$$S_i = 2 \frac{\partial W}{\partial C} \quad (4.23)$$

and the large-deformation stress update for the Maxwell element stresses is given by:

$$S'_i(t_1) = S'_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^G}\right) + \alpha_i^G \exp\left(-\frac{\Delta t}{2\tau_i^G}\right) \left[\frac{dw}{dC} \Big|_{t_1} - \frac{dw}{dC} \Big|_{t_0} \right] \quad (4.24)$$

$$p_i(t_1) = p_i(t_0) \exp\left(-\frac{\Delta t}{\tau_i^K}\right) + \alpha_i^K \exp\left(-\frac{\Delta t}{2\tau_i^K}\right) \left[\frac{dU}{dJ} \Big|_{t_1} - \frac{dU}{dJ} \Big|_{t_0} \right] \quad (4.25)$$

where:

S'_i = deviatoric component of S_i

p_i = pressure component of S_i

An anisotropic hyperelastic model can also be used for [Equation 4.22](#) (p. 100), in which case the form of the Maxwell element stress updates are unchanged.

This model requires the Prony series parameters to be input via the Prony data table (as described in [Small Deformation](#) (p. 97)). The hyperelastic parameters for this model are input via a hyperelastic data table (**TB, HYPER**). For more information, see [Hyperelasticity](#) (p. 83).

4.7.1.4. Stored Energy and Dissipation

For a physical interpretation of the Prony series formulation, the stored energy is the potential energy of the spring elements and the dissipated energy is the energy used to deform the dashpots in the Maxwell elements.

The increment of energy used by the dashpots over a time increment is:

$$\Delta w_d = \sum_{i=1}^{n_g} \dot{s}_i \cdot \delta e_i^\eta dt + \sum_{i=1}^{n_k} \dot{p}_i \cdot \delta \Delta_i^\eta dt \quad (4.26)$$

where, for dashpot i , $\bar{s}_i$ is the deviatoric stress, δe_i^η is the increment in deviatoric strain, $\bar{p}_i$ is the pressure, and $\delta \Delta_i^\eta$ is the increment in volumetric strain.

For an additive decomposition of the Maxwell element strains, the dashpot strain increment is given by:

$$\Delta \epsilon_i^\eta = \Delta \epsilon - C_i : \Delta \sigma_i$$

where $\Delta \epsilon$ is the strain increment, C_i is the Maxwell spring compliance, and $\Delta \sigma_i$ is the stress increment. Defining C_0 as the instantaneous compliance, the Maxwell spring compliance is:

$$C_i = \frac{1}{\alpha_i} C_0$$

The dashpot strain increment and compliance can be applied to both deviatoric and volumetric components, and then α_i corresponds to either the deviatoric or volumetric relative modulus for Maxwell element i .

Using Equation 4.26 (p. 100), the stored energy increment is given by the total energy increment minus the dissipation:

$$\Delta w_s = \Delta W - \Delta w_d$$

where W is the total energy and the increment is given by:

$$\Delta W = \sigma : \Delta \epsilon$$

Used in the automatic time-stepping (**CUTCONTROL**), a scalar equivalent viscous-strain increment can be defined as:

$$\delta \epsilon_{eq}^\eta = \left(\frac{2}{3} \delta e^\eta \cdot \delta e^\eta + \frac{1}{3} \delta \Delta^\eta \right)^{1/2}$$

where:

$$\begin{aligned} \delta e^\eta &= \sum_{i=1}^{n_g} \alpha_i^G \delta e_i^\eta \\ \delta \Delta^\eta &= \sum_{i=1}^{n_k} \alpha_i^K \delta \Delta_i^\eta \end{aligned}$$

For large deformation, the incremental stored energy and dissipation use strain increments calculated from the rate-of-deformation tensor and a Maxwell-compliance tensor calculated from the instantaneous hyperelastic stiffness and the relative moduli.

The viscoelastic stored- and dissipated-energy results are stored as item SEND and components ELASTIC and VDAM, respectively.

4.7.2. Time-Temperature Superposition

For thermorheologically simple materials, the influence on the material behavior due to changing temperature is the same as that due to changing time. For these materials, a rate-dependent material response, P (a function of temperature and time), can be reduced to:

$$P(T, t) = P(T_r, \xi) \quad (4.27)$$

where:

T = current absolute temperature

T_r = constant absolute reference temperature

ξ = shifted time given by $\xi = A(T) \cdot t$, where $A(T)$ = shift function.

The constitutive equations are solved in the shifted time scale. This method has the potential to reduce the experimental effort required to determine the material parameters but requires the determination of the shift function.

The shift functions, $A(T)$, are evaluated in an absolute temperature scale determined by adding the temperature offset value (**TOFFST**) to the current temperature, reference temperature, and fictive temperature in the shift functions.

The following forms of the shift function are available:

4.7.2.1. Williams-Landel-Ferry Shift Function

4.7.2.2. Tool-Narayanaswamy Shift Function

4.7.2.3. User-Defined Shift Function

4.7.2.1. Williams-Landel-Ferry Shift Function

The Williams-Landel-Ferry shift function has the form:

$$\log_{10}(A(T)) = \frac{C_1(T - T_r)}{C_2 + (T - T_r)} \quad (4.28)$$

where C_1 and C_2 are material parameters. (The shift function is often given in the literature with the opposite sign.)

The parameters are input via a shift function data table (**TB,SHIFT**).

For the Williams-Landel-Ferry shift function, $TB0PT = WLF$, and the required input constants are:

Table Location	Constant
1	T_r
2	C_1
3	C_2

4.7.2.2. Tool-Narayanaswamy Shift Function

Two forms of the Tool-Narayanaswamy shift function are available, one of which includes a fictive temperature.

The first form is given by:

$$\ln(A(T)) = \frac{H}{R} \left(\frac{1}{T_r} - \frac{1}{T} \right) \quad (4.29)$$

where $\frac{H}{R}$ is the scaled activation energy.

The parameters are input in a shift function data table (**TB,SHIFT**).

For the Tool-Narayanaswamy shift function, $TBOPT = TN$, and the required input constants are:

Table Location	Constant
1	T_r
2	$\frac{H}{R}$

The second form of the Tool-Narayanaswamy shift function includes an evolving fictive temperature. The fictive temperature is used to model material processes that contain an intrinsic equilibrium temperature that is different from the ambient temperature of the material. The shift function is given by:

$$\ln(A(T)) = \frac{H}{R} \left(\frac{1}{T_r} - \frac{X}{T} - \frac{1-X}{T_F} \right) \quad (4.30)$$

where:

X = weight parameter

T_F = absolute fictive temperature.

The partial fictive temperatures are calculated in the relative temperature scale defined by the input material parameters. The evolving fictive temperature is given by:

$$T_F = \sum_{i=1}^{n_f} C_{fi} T_{fi} \quad (4.31)$$

where:

n_f = number of partial fictive temperatures

C_{fi} = fictive temperature relaxation coefficient

T_{fi} = partial fictive temperature

The evolution of the partial fictive temperature is given by:

$$T_{fi} = \frac{\tau_{fi} T_{fi}^0 + T \Delta t A(T_F^0)}{\tau_{fi} + \Delta t A(T_F^0)} \quad (4.32)$$

where:

τ_{fi} = fictive temperature relaxation time

0 (superscript) = values from the previous time step

The fictive temperature model modifies the volumetric thermal strain model and gives an incremental thermal strain as:

$$\Delta \varepsilon^T = \alpha_g(T) \Delta T + [\alpha_l(T_F) - \alpha_g(T_F)] \Delta T_F \quad (4.33)$$

where:

ΔT = temperature increment over the time step. The temperature increment in the first increment is the body temperature at the end of the increment minus the fictive thermal strain reference temperature, T_{ref} , defined in the shift function table. If T_{ref} is 0 or undefined in the shift function table, the shift function reference temperature, T_r , is used to calculate the temperature increment in the first time step.

α_g and α_l = glass and liquid coefficients, respectively, of thermal expansion given by:

$$\alpha_g(T) = \alpha_{g0} + \alpha_{g1}T + \alpha_{g2}T^2 + \alpha_{g3}T^3 + \alpha_{g4}T^4 \quad (4.34)$$

$$\alpha_l(T) = \alpha_{l0} + \alpha_{l1}T + \alpha_{l2}T^2 + \alpha_{l3}T^3 + \alpha_{l4}T^4 \quad (4.35)$$

Equation 4.34 (p. 104) and Equation 4.35 (p. 104) are evaluated at the relative current and fictive temperatures.

The parameters are input in a shift function data table (**TB,SHIFT**).

For the Tool-Narayanaswamy with fictive temperature shift function, $TBOPT = FICT$, $NPTS = n_f$, and the required input constants are:

Table Location	Constant
1	T_r
2	H/R
3	X
4 to 3(NPTS + 1)	$T_{f1}, C_{f1}, \tau_{f1}, T_{f2}, C_{f2}, \tau_{f2}, \dots, T_{fn}, C_{fn}, \tau_{fn}$
3(NPTS + 1) + 1 to 3(NPTS + 1) + 5	$\alpha_{g0}, \alpha_{g1}, \alpha_{g2}, \alpha_{g3}, \alpha_{g4}$
3(NPTS + 1) + 6 to 3(NPTS + 1) + 10	$\alpha_{l0}, \alpha_{l1}, \alpha_{l2}, \alpha_{l3}, \alpha_{l4}$
3(NPTS + 1) + 11	T_{ref}

4.7.2.3. User-Defined Shift Function

Other shift functions can be accommodated via the user-provided subroutine **UsrShift**, described in the [Programmer's Reference](#).

Given the input parameters, the routine must evolve the internal state variables, then return the current and half-step shifted time.

4.7.3. Harmonic Viscoelasticity

For use in [harmonic analyses](#), the generalized Maxwell model can be used to provide a constitutive model in the harmonic domain.

Assuming that the strain varies harmonically and that all transient effects have subsided, Equation 4.12 (p. 97) has the form:

$$\sigma = 2[G' + iG'']e^{\exp(i(\Omega t + \delta))} + [K' + iK'']\Delta \exp(i(\Omega t + \delta)) \quad (4.36)$$

where:

e, Δ = deviatoric and volumetric components of strain

G', G'' = storage and loss shear moduli

K', K'' = storage and loss bulk moduli

Ω, δ = frequency and phase angle

Comparing Equation 4.36 (p. 104) to the harmonic equation of motion, the material stiffness is due to the storage moduli and the material damping matrix is due to the loss moduli divided by the frequency.

The following additional topics for harmonic viscoelasticity are available:

[4.7.3.1. Prony Series Complex Modulus](#)

[4.7.3.2. Experimental Data Complex Modulus](#)

[4.7.3.3. Frequency-Temperature Superposition](#)

[4.7.3.4. Stress](#)

4.7.3.1. Prony Series Complex Modulus

The storage and loss moduli are related to the Prony parameters by:

$$G' = G_0 \left[1 - \sum_{i=1}^{n_G} \left(\alpha_i^G - \frac{\alpha_i^G (\tau_i^G \Omega)^2}{1 + (\tau_i^G \Omega)^2} \right) \right], G'' = G_0 \sum_{i=1}^{n_G} \left(\frac{\alpha_i^G \tau_i^G \Omega}{1 + (\tau_i^G \Omega)^2} \right) \quad (4.37)$$

$$K' = K_0 \left[1 - \sum_{i=1}^{n_K} \left(\alpha_i^K - \frac{\alpha_i^K (\tau_i^K \Omega)^2}{1 + (\tau_i^K \Omega)^2} \right) \right], K'' = K_0 \sum_{i=1}^{n_K} \left(\frac{\alpha_i^K \tau_i^K \Omega}{1 + (\tau_i^K \Omega)^2} \right) \quad (4.38)$$

Input of the Prony series parameters for a viscoelastic material in harmonic analyses follows the input method for viscoelasticity in the time domain detailed above.

4.7.3.2. Experimental Data Complex Modulus

Storage and loss moduli can also be input as piecewise linear functions of frequency $f = \Omega / 2\pi$ on a data table for [experimental data](#) (p. 26). Isotropic elastic moduli can be input for the complex shear, bulk and tensile modulus as well as the complex Poisson's ratio.

The points for the experimental data table (input via the **TBPT** command) have frequency as the independent variable, and the dependent variables are the real component, imaginary component, and $\tan(\delta)$. If the imaginary component is empty or zero for the data point, the $\tan(\delta)$ value is used to determine it; otherwise $\tan(\delta)$ is not used.

Input complex shear modulus on an experimental data table (**TB,EXPE**) with **TBOPT** = GMODULUS. The data points are defined by:

Position	Value
1	f
2	G'
3	G''

Position	Value
4	$\tan(\delta) = \frac{G''}{G'}$

Input complex bulk modulus on an experimental data table (**TB**,EXPE) with *TBOPT* = KMODULUS. The data points are defined by:

Position	Value
1	f
2	K'
3	K''
4	$\tan(\delta) = \frac{K''}{K'}$

Input complex tensile modulus on an experimental data table (**TB**,EXPE) with *TBOPT* = EMODULUS. The data points are defined by:

Position	Value
1	f
2	E'
3	E''
4	$\tan(\delta) = \frac{E''}{E'}$

Input complex Poisson's ratio on an experimental data table (**TB**,EXPE) with *TBOPT* = NUXY. The data points are defined by:

Position	Value
1	f
2	ν'
3	ν''
4	$\tan(\delta) = \frac{\nu''}{\nu'}$

Using experimental data to define the complex constitutive model requires elastic constants (defined via **MP** or by an elastic data table [**TB**,ELASTIC]). The elastic constants are unused if two sets of complex modulus experimental data are defined. This model also requires an empty Prony data table (**TB**,PRONY) with *TBOPT* = EXPERIMENTAL.

Two elastic constants are required to define the complex constitutive model. If only one set of experimental data for a complex modulus is defined, the Poisson's ratio (defined via **MP** or by elastic data table) is used as the second elastic constant.

4.7.3.3. Frequency-Temperature Superposition

For thermorheologically simple materials in the frequency domain, frequency-temperature superposition is analogous to using time-temperature superposition to shift inverse frequency. The Williams-Landel-Ferry and Tool-Narayanaswamy (without fictive temperature) shift functions can be

used in the frequency domain, and the material parameter input follows the shift table input described in [Time-Temperature Superposition \(p. 102\)](#).

Frequency-temperature superposition can be used with either the Prony series complex modulus or any of the experimental data for complex moduli or Poisson's ratio.

4.7.3.4. Stress

The magnitude of the real and imaginary stress components are obtained from expanding [Equation 4.36 \(p. 104\)](#) and using the storage and loss moduli from either the Prony series parameters or the experimental data:

$$\operatorname{Re}(\sigma) = 2(G'e' - G''e'') + I(K'\Delta' - K''\Delta'') \quad (4.39)$$

$$\operatorname{Im}(\sigma) = 2(G'e'' + G''e') + I(K'\Delta'' + K''\Delta') \quad (4.40)$$

where:

$\operatorname{Re}(\sigma)$ = real stress magnitude

$\operatorname{Im}(\sigma)$ = imaginary stress magnitude

4.8. Microplane

The microplane model (**TB**,MPLANE) is based on research by Bazant and Gambarova [1][2] in which the material behavior is modeled through stress-strain laws on a number of individual planes.

Directional-dependent stiffness degradation is modeled through damage laws on individual potential failure planes, leading to a macroscopic anisotropic damage formulation.

The model is well suited for simulating engineering materials consisting of various aggregate compositions with differing properties (for example, concrete modeling, in which rock and sand are embedded in a weak matrix of cements).

The following microplane material model types are available:

- [Elastic damage \(p. 111\)](#) ([nonregularized \(p. 112\)](#) and [regularized \(p. 113\)](#) forms)
- [Coupled damage-plasticity \(p. 116\)](#) (regularized form only)

The microplane model cannot be combined with other material models.

The following microplane model topics are available:

[4.8.1. Microplane Modeling](#)

[4.8.2. Microplane Material Models](#)

[4.8.3. Learning More About Microplane Material Modeling](#)

Also see [Material Model Element Support \(p. 5\)](#) for microplane.

4.8.1. Microplane Modeling

Three primary tasks summarize microplane theory:

1. Apply a kinematic constraint to relate the macroscopic strain tensors to their microplane counterparts (projection).
2. Define the constitutive laws on the microplane levels, where constitutive equations are applied on each microplane.
3. Relate the homogenization process on the material point level to derive the overall material response. (Homogenization is based on the principle of energy equivalence.)

The microplane material model formulation is based on the assumption that microscopic free energy Ψ^{mic} on the microplane level exists and that the integral of Ψ^{mic} over all microplanes is equivalent to a macroscopic free Helmholtz energy Ψ^{mac} [3], expressed as:

$$\psi^{mac} = \frac{3}{4\pi} \int_{\Omega} \psi^{mic} d\Omega$$

The factor $\frac{3}{4\pi}$ results from the integration of the sphere of unit radius with respect to the area Ω .

The strains and stresses at microplanes are additively decomposed into volumetric and deviatoric parts, respectively, based on the volumetric-deviatoric (V-D) split.

The strain split is expressed as:

$$\varepsilon = \varepsilon_D + \varepsilon_v \mathbf{1}$$

The scalar microplane volumetric strain ε_v results from:

$$\varepsilon_v = \mathbf{V} : \varepsilon \text{ where } \mathbf{V} = \frac{1}{3} \mathbf{1}$$

where $\mathbf{V}$ is the second-order volumetric projection tensor and $\mathbf{1}$ the second-order identity tensor.

The deviatoric microplane strain vector ε_D is calculated as:

$$\varepsilon_D = Dev : \varepsilon \text{ and } Dev = \mathbf{n} \cdot \mathbf{\Pi} - \frac{1}{3} \mathbf{n} \cdot \mathbf{1} \otimes \mathbf{1} = \mathbf{n} \cdot \mathbf{\Pi}^{dev}$$

where $\mathbf{\Pi}$ is the fourth-order symmetric identity tensor and the vector $\mathbf{n}$ describes the normal on the microsphere (microplane).

The stresses can then be derived by taking the free energy derivative with respect to the strain tensor:

$$\sigma = \frac{3}{4\pi} \int_{\Omega} \frac{\partial \psi^{mic}}{\partial \varepsilon} d\Omega = \frac{3}{4\pi} \int_{\Omega} (\mathbf{V} \sigma_v + 2 Dev^T \cdot \sigma_D) d\Omega$$

where σ_v and σ_D are the scalar volumetric stress and the deviatoric stress tensor on the microsphere, and $\sigma_v = \mathbf{V} : \sigma$ and $\sigma_D = Dev : \sigma$.

Assume isotropic elasticity:

$$\sigma_v = \frac{\partial \psi^{mic}}{\partial \varepsilon_v} = K^{mic} \varepsilon_v$$

and

$$\sigma_D = \frac{\partial \psi^{mic}}{\partial \varepsilon_D} = G^{mic} \varepsilon_D$$

where K^{mic} and G^{mic} are microplane elasticity parameters and can be interpreted as a sort of microplane bulk and shear modulus. They are related to the elastic macroscopic parameters as follows:

$$K^{mic} = 3K$$

$$G^{mic} = G$$

Integration over the surface of the sphere in order to calculate the homogenized quantities is achieved by numerical integration:

$$\frac{3}{4\pi} \int_{\Omega} \cdot^{mic} d\Omega = \sum_{i=1}^{N_p} (\cdot^{mic})^i w^i$$

where w^i is the weight factor.

The following additional topics for microplane modeling are available:

[4.8.1.1. Discretization](#)

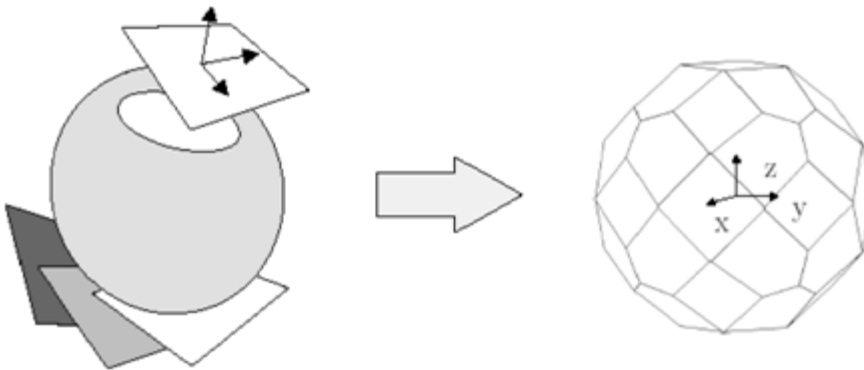
[4.8.1.2. Implicit Gradient Regularization](#)

4.8.1.1. Discretization

Discretization is the transfer from the microsphere to microplanes which describe the approximate form of the sphere. Forty-two microplanes are used for the numerical integration. Due to the symmetry of the microplanes (where every other plane has the same normal direction), 21 microplanes are considered. [3]

The following figure illustrates the discretization process:

Figure 4.23: Sphere Discretization by 42 Microplanes



4.8.1.2. Implicit Gradient Regularization

Strain-softening material models often cause mesh sensitivity and numerical instability, a problem mitigated by implicit gradient regularization, a class of nonlocal methods. Implicit gradient regularization enhances a local variable by considering its nonlocal counterpart as an extra degree of freedom governed by a Helmholtz-type equation.

The governing equations are therefore given by the linear momentum-balance equation and a modified Helmholtz equation describing the nonlocal equivalent strain field $\tilde{\eta}_m$:

$$\begin{aligned}\nabla \cdot \sigma + f &= 0 \\ \tilde{\eta}_m - c \nabla^2 \tilde{\eta}_m &= \eta_m\end{aligned}$$

where σ is the Cauchy stress tensor, f is the body force vector, $\nabla \cdot$ is the divergence, ∇ is the gradient, and ∇^2 is the Laplace operator. The gradient parameter c controls the [range of nonlocal interaction](#) (p. 110). The equivalent strain η_m is the local variable to be enhanced, and $\tilde{\eta}_m$ is its nonlocal counterpart.

The homogeneous Neumann boundary condition is used as follows:

$$\nabla \tilde{\eta}_m \cdot n_b = 0$$

where n_b is the normal to the outer boundary of the nonlocal field. With the homogeneous Neumann boundary condition, no explicit definition of boundary conditions for the extra degrees of freedom is required.

The [elastic damage microplane material model](#) (p. 111) is available in [nonregularized](#) (p. 112) and [regularized](#) (p. 113) forms. The [coupled damage-plasticity microplane model](#) (p. 116) is available in a regularized form only.

4.8.1.2.1. Identifying the Nonlocal Interaction-Range Parameter c

Identifying the nonlocal interaction-range parameter c (related to the length-scale parameter l by the equation $c=l^2$) can be somewhat challenging.

One method [8] uses the results comparison of homogeneous and nonhomogeneous tensile tests of concrete to identify the parameter. The first experiment consists of a specimen restrained such that the cracking is distributed, while the second consists of a notched specimen to trigger a localized failure. The damage parameters can be obtained from homogeneous experiments because, in this case, the damage is uniformly distributed along the whole specimen and is unaffected by the nonlocal parameter c . In the first step, therefore, the damage parameters are identified from the stress-strain curve for the homogeneous test. In the second step, the force-displacement curve for the notched-specimen test identifies the nonlocal parameter c , keeping the damage parameters constant.

Other approaches to determine the gradient parameter c use the size effects observed in concrete and inverse calibration of force-deflection curves [9], and calibration based on fracture-energy test and measurement of crack-surface roughness [10].

4.8.2. Microplane Material Models

The following microplane material model topics are available:

[4.8.2.1. Elastic Damage Microplane Material Models](#)

[4.8.2.2. Coupled Damage-Plasticity Microplane Model](#)

[4.8.2.3. Solution Control for the Regularized Microplane Damage Models](#)

4.8.2.1. Elastic Damage Microplane Material Models

To account for material degradation and damage, the microscopic free-energy function is modified to include a damage parameter, yielding:

$$\psi^{mic}(\varepsilon_v, \varepsilon_D, d^{mic}) = (1 - d^{mic}) \psi^{mic}(\varepsilon_v, \varepsilon_D)$$

The damage parameter d^{mic} is the normalized damage variable $\{0 \leq d^{mic} \leq 1\}$.

The stresses are derived by:

$$\sigma = \frac{3}{4\pi} \int_{\Omega} \frac{\partial \psi^{mic}}{\partial \varepsilon} d\Omega = \frac{3}{4\pi} \int_{\Omega} (\mathbf{v} \sigma_v + 2 \text{Dev}^T \cdot \sigma_D) d\Omega$$

where $\sigma_D(\varepsilon_D, d^{mic}) = (1 - d^{mic}) G^{mic} \varepsilon_D$ and $\sigma_v(\varepsilon_v, d^{mic}) = (1 - d^{mic}) K^{mic} \varepsilon_v$. The stresses derived generally result in unsymmetric material stiffness; therefore, use the unsymmetric Newton-Raphson solver (**NROPT**, **UNSYM**) in such cases to facilitate convergence.

The damage status of a material is described by the equivalent-strain-based damage function $\Phi^{mic} = \varphi(\eta^{mic}) - d^{mic} \leq 0$, where η^{mic} is the *equivalent strain*, which characterizes the damage evolution law and is defined as:

$$\eta^{mic} = k_0 I_1 + \sqrt{k_1^2 I_1^2 + k_2 J_2}$$

where I_1 is the first invariant of the strain tensor ε , J_2 is the second invariant of the deviatoric part of the strain tensor ε , and k_0 , k_1 , and k_2 are material parameters that characterize the form of damage function. The [equivalent strain function \(p. 111\)](#) implies the Mises-Hencky-Huber criterion for $k_0 = k_1 = 0$, and $k_2 = 1$, and the Drucker-Prager-criterion for $k_0 > 0$, $k_1 = 0$, and $k_2 = 1$.

The damage evolution is modeled by the following function:

$$d^{mic} = 1 - \frac{\gamma_0^{mic}}{\gamma^{mic}} \left[1 - \alpha^{mic} + \alpha^{mic} \exp\left(\beta^{mic} (\gamma_0^{mic} - \gamma^{mic})\right) \right]$$

where α^{mic} defines the maximal degradation, β^{mic} determines the rate of damage evolution, and γ_0^{mic} is the damage threshold which characterizes the equivalent strain on which the material damaging starts (damage starting boundary).

γ^{mic} is a history variable representing the largest value of equivalent strain in the material's history. The variable is defined differently depending on whether it is the nonregularized or the regularized version of the elastic damage microplane material model. Theoretically, the history variable definition is how the two versions differ.

For postprocessing, the maximum damage d^{max} is defined as the maximum value of microplane damage values d^{mic} . The macroscopic damage is defined by:

$$d^{mac} = \frac{1}{4\pi} \int_{\Omega} d^{mic} d\Omega$$

and has a range of $0 \leq d^{mac} \leq 1$.

Both [nonregularized](#) (p. 112) and [regularized](#) (p. 113) versions of the elastic damage microplane material model are available.

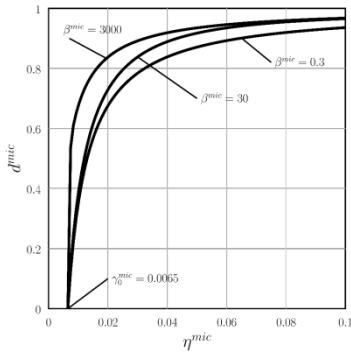
4.8.2.1.1. Nonregularized Elastic Damage Microplane Material Model

The history variable γ^{mic} is calculated as the maximum of the damage threshold γ_0^{mic} and the equivalent strain $\bar{\eta}^{mic}$ for each microplane:

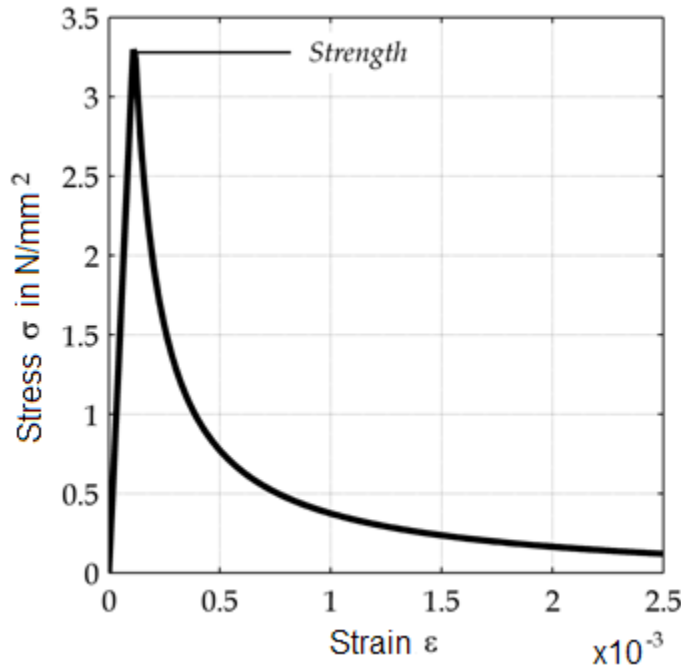
$$\gamma^{mic} = \max(\gamma_0^{mic}, \bar{\eta}^{mic})$$

This figure shows the evolution of the damage variable d^{mic} as a function of equivalent strain η^{mic} for the implemented exponential damage model:

Figure 4.24: Damage Parameter d^{mic} Depending on the Equivalent Strain



This figure shows the stress-strain behavior for uniaxial tension:

Figure 4.25: Stress-strain Behavior at Uniaxial Tension

4.8.2.1.2. Regularized Elastic Damage Microplane Material Model

Use the regularized elastic damage microplane model, based on the research by Zreid and Kaliske [6], to overcome the numerical instability and pathological mesh sensitivity to which the nonregularized version of the model is susceptible. The regularized model uses an implicit gradient regularization scheme, defined via a nonlocal field, that adds one extra degree of freedom per node.

The history variable γ^{mic} is calculated as the maximum of the damage threshold γ_0^{mic} and the equivalent strain $\bar{\eta}_{mic}$ for each microplane:

$$\gamma^{mic} = \max(\gamma_0^{mic}, \bar{\eta}^{mic})$$

The modified equivalent strains for each microplane are calculated via implicit gradient regularization.

Implicit Gradient Regularization

The governing equations and boundary condition for the regularized microplane models are included again here for completeness:

$$\nabla \cdot \sigma + f = 0$$

$$\bar{\eta}_m - c \nabla^2 \bar{\eta}_m = \eta_m$$

$$\nabla \bar{\eta}_m \cdot n_b = 0$$

where η_m is the equivalent strain of a given microplane and $\bar{\eta}_m$ is its nonlocal counterpart. The microplane chosen for η_m is the one having the largest equivalent strain $\eta_m = \max_{mic=1}^{21}(\eta^{mic})$.

The nonlocal counterpart $\bar{\eta}_m$ is obtained as part of solving the system of governing equations shown above.

The equivalent strains of each microplane can now be calculated by modifying them with a ratio of local to nonlocal largest equivalent strain:

$$\bar{\eta}^{mic} = \frac{\bar{\eta}_m}{\eta_m} \eta^{mic}$$

For more information, see [Implicit Gradient Regularization](#) (p. 110).

4.8.2.1.3. Defining the Elastic Damage Microplane Material Models

The [nonregularized](#) (p. 112) elastic damage microplane model requires eight parameters, while the [regularized](#) (p. 113) elastic damage microplane model requires nine parameters.

The following elasticity and damage parameters are common to both models and are defined in the same way for both:

Parameter Type	Parameter	Description	Model	
			Nonregularized	Regularized
Elasticity	E	Modulus of elasticity	Y	Y
	ν	Poisson's ratio	Y	Y
Damage	k_0, k_1, k_2	Damage function material parameters	Y	Y
	γ_0^{mic}	Damage threshold	Y	Y
	α^{mic}	Maximum damage parameter	Y	Y
	β^{mic}	Rate of damage evolution	Y	Y
Nonlocal	c	Nonlocal interaction range parameter	N	Y

Define the elastic parameters E and ν via **TB,ELAS** or **MP**.

Define the damage parameters via **TB,MPLANE,,,ORTH**.

The following table describes the damage material constants:

Constant	Meaning
C1	k_0
C2	k_1
C3	k_2
C4	γ_0^{mic}

Constant	Meaning
C5	α^{mic}
C6	β^{mic}

Example 4.21: Microplane Material Constant Input

```
! Define elastic properties of material
tb,elas,1
tbdata,1,60000.0,0.36

! Define microplane model properties
tb,mplane,1,,6,orth
tbdata,1,0,0,1,0.1,0.1,0.1
```

Considerations for Defining the Regularized Model:

- Elements supported: [CPT212](#), [CPT213](#), [CPT215](#), [CPT216](#), and [CPT217](#).
- To activate the required extra degree of freedom (GFV1), set KEYOPT(18) = 1. The extra degree of freedom requires no boundary-condition input.
- A fine mesh is recommended, particularly at probable damage-prone regions. To observe mesh-independent results, the mesh size may need to be less than half the square root of the nonlocal parameter.
- Define the nonlocal parameter via **TB,MPLANE,,,,NLOCAL**. Following is the material data-table constant:

Constant	Meaning
C1	c

Example 4.22: Regularized Elastic Damage Microplane Material Constant Input

```
/prep7
! Define element
ET,1,215
KEYO,1,18,1 ! Activate extra degree of freedom

! Parameter values
E=18000
nu=.18

k0=0.703125
k1=0.703125
k2=0.2154553289284690
gamma=.0002
alpha=.96
beta=450
c=25

! Define elastic properties of material
mp,ex,1,E
mp,nuxy,1,nu

! Define microplane model properties
tb,mplane,1,,,ORTH
TBDATA,1,k0,k1,k2,gamma,alpha,beta

tb,mplane,1,,,NLOCAL
TBDATA,1,c
```

4.8.2.1.4. Output for the Elastic Damage Microplane Material Models

Some microplane element quantities are available for postprocessing (/POST1, /POST26). The assigned label is MPLA.

Element outputs include the homogenized damage (DMAC), and the maximum microplane damage (DMAX).

4.8.2.2. Coupled Damage-Plasticity Microplane Model

Use this model, based on research by Zreid and Kaliske [4][5][6], to overcome the numerical instability and pathological mesh sensitivity to which strain-softening materials such as the microplane model are susceptible.

The model uses an implicit gradient regularization scheme, defined via a nonlocal field, that adds two extra degrees of freedom per node. Microplane plasticity is also introduced, using microplane quantities, through laws resembling classical invariant-based plasticity models, enabling material models with a direct link to the conventional macroscopic plasticity models.

The plasticity in this model is defined via a three-surface microplane Drucker-Prager model, covering a full range of possible stress states and enabling cyclic loading. The damage includes a tension-compression split to account for transition of the stress state during cyclic loading.

To account for coupled damage-plasticity, the microscopic free-energy function is once again modified to include damage; furthermore, the total strain components are additively decomposed into elastic and plastic parts. The resulting stress-strain relation is:

$$\sigma = \frac{3}{4\pi} \int_{\Omega} (1 - d^{mic}) \left[K^{mic} V(\varepsilon_V - \varepsilon_V^{pl}) + 2G^{mic} Dev^T(\varepsilon_D - \varepsilon_D^{pl}) \right] d\Omega$$

where d^{mic} is the normalized damage variable $\{0 \leq d^{mic} \leq 1\}$, ε_V^{pl} is the volumetric microplane plastic strain, and ε_D^{pl} is the deviatoric plastic strain.

The microplane plastic strain rate evolutions are governed by the following flow rules:

$$\dot{\varepsilon}_V^{pl} = \dot{\lambda} \frac{\partial f^{mic}}{\partial \sigma_V^e}$$

$$\dot{\varepsilon}_D^{pl} = \dot{\lambda} \frac{\partial f^{mic}}{\partial \sigma_D^e}$$

where $\dot{\lambda}$ is the plastic multiplier, and f^{mic} is the given microplane yield function.

The microplane volumetric and deviatoric effective stresses, σ_V^e and σ_D^e respectively, are defined as:

$$\sigma_V^e = K^{mic} (\varepsilon_V - \varepsilon_V^{pl})$$

$$\sigma_D^e = 2G^{mic} (\varepsilon_D - \varepsilon_D^{pl})$$

The following additional topics about the coupled damage-plasticity microplane model are available:

4.8.2.2.1. Smooth Three-Surface Microplane Drucker-Prager Cap Yield Function

4.8.2.2.2. Damage Evolution

4.8.2.2.3. Implicit Gradient Regularization

4.8.2.2.4. Coupled Damage-Plasticity Microplane Model Parameters

4.8.2.2.5. Defining the Coupled Damage-Plasticity Microplane Model

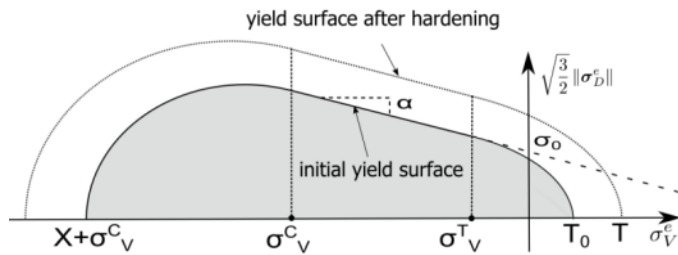
4.8.2.2.6. Identifying Coupled Damage-Plasticity Microplane Model Parameters

4.8.2.2.7. Coupled Damage-Plasticity Microplane Damage Output

4.8.2.2.1. Smooth Three-Surface Microplane Drucker-Prager Cap Yield Function

A smooth Drucker-Prager yield function with tension and compression caps covers the material response under all possible triaxialities:

Figure 4.26: Smooth Three-Surface Microplane Cap Yield Function



The Drucker-Prager cap model is specific to microplane plasticity, as it uses microplane quantities. It is similar but not identical to the macroscopic [Extended Drucker-Prager Cap \(p. 60\)](#) model. The yield function is expressed as:

$$f^{mic} = \frac{3}{2} \sigma_D^e \cdot \sigma_D^e - f_1^2 f_c f_t$$

where f_1 is the Drucker-Prager yield function with hardening, f_c is the compression cap, and f_t is the tension cap.

The yield function is evaluated in the undamaged stress space. The product $f_1^2 f_c$ ensures that the compression cap has the same slope as the function f_1 at the intersection point between f_1 and f_c . The same occurs for the tension cap, and so overall the yield surface has C^1 continuity.

The Drucker-Prager yield function f_1 is defined as:

$$f_1 = \sigma_0 - \alpha \sigma_V^e + f_h$$

where σ_0 is the initial yield stress, α is a friction coefficient, and f_h is a hardening function.

The compression cap is defined by:

$$f_c = 1 - H(\sigma_V^C - \sigma_V^e) \frac{(\sigma_V^C - \sigma_V^e)^2}{X^2}$$

$$X = R f_1(\sigma_V^C)$$

where $H(\cdot)$ is the Heaviside step function, and σ_V^C is the abscissa of the intersection point between f_c and f_1 . R is the ratio between the major (deviatoric) and minor (volumetric) axes of the cap. The Heaviside function is used to activate the cap only when the stress state is in its domain.

The tension cap is defined by:

$$f_t = 1 - H(\sigma_V^e - \sigma_V^T) \frac{(\sigma_V^e - \sigma_V^T)^2}{(T - \sigma_V^T)^2}$$

$$T = T_0 + R_T f_h$$

where σ_V^T is the abscissa of the intersection point between f_1 and f_t . T_0 is the initial intersection of the cap with the volumetric axis, and R_T is a hardening parameter controlling the increase of the intersection point T due to hardening.

The hardening is considered to be linear and is defined by:

$$f_h = D\kappa$$

$$\dot{\kappa} = \dot{\lambda}$$

where D is a hardening parameter, and κ is a hardening variable.

4.8.2.2.2. Damage Evolution

The damage evolution behavior is motivated by the material behavior of concrete and similar materials.

To realistically model the damage of concrete subject to cyclic loading, the following considerations are taken into account:

- **The initiation of damage and its subsequent evolution is different between compression and tension.**
 - Concrete is more brittle in tension, and softening begins to occur almost immediately after the elastic limit.
 - In compression, some hardening is observed after the elastic limit before softening occurs.
- **In the transition from tension to compression states, the stiffness lost during tensile cracking is recovered due to crack closure. Upon transition to tension, however, the damage sustained under compression is retained.**

This unique behavior is described via a damage split, where the total damage d^{mic} is decomposed into compression d_c^{mic} and tension d_t^{mic} parts, as follows:

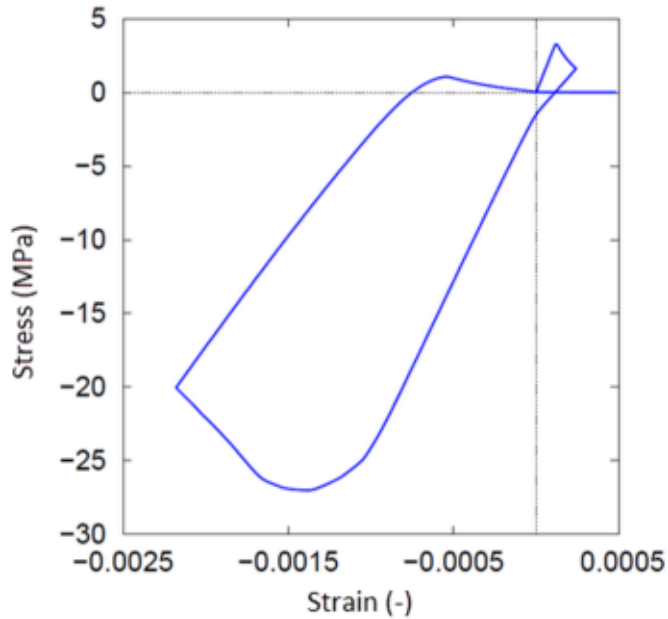
$$1 - d^{mic} = (1 - d_c^{mic})(1 - r_w d_t^{mic})$$

$$r_w = \frac{\sum_{i=1}^3 \langle \epsilon^i \rangle}{\sum_{i=1}^3 |\epsilon^i|}$$

where r_w is the split weight factor, ε^I represents the principal strain values, and $\langle \varepsilon^I \rangle$ indicates the positive principal strain values.

This figure shows the stiffness reduction in unloading and the stiffness recovery in compression:

Figure 4.27: Effect of Damage Split on Tension-Compression Cyclic Loading Response



The damage laws are defined as:

$$d_t^{mic} = 1 - \exp(-\beta_t \gamma_t^{mic})$$

$$d_c^{mic} = 1 - \exp(-\beta_c \gamma_c^{mic})$$

where β_t and β_c are material constants. γ_t^{mic} and γ_c^{mic} , the variables driving the damage, are calculated from over-nonlocal equivalent strains (p. 120).

For postprocessing, the homogenized damage d^{hom} is defined by:

$$d^{hom} = \frac{1}{4\pi} \int_{\Omega} d^{mic} d\Omega$$

Equivalent strain rates $\dot{\eta}_t^{mic}$ and $\dot{\eta}_c^{mic}$ are functions of the volumetric plastic strain rate, as follows:

$$\dot{\eta}_t^{mic} = \begin{cases} r_w \dot{\varepsilon}_V^{pl} & \dot{\varepsilon}_V^{pl} > 0 \\ 0 & \dot{\varepsilon}_V^{pl} \leq 0 \end{cases}$$

$$\dot{\eta}_c^{mic} = \begin{cases} (1-r_w) \dot{\varepsilon}_V^{pl} & \dot{\varepsilon}_V^{pl} > 0 \\ 0 & \dot{\varepsilon}_V^{pl} \leq 0 \end{cases}$$

4.8.2.2.3. Implicit Gradient Regularization

The governing equations and boundary condition are included again here for completeness:

$$\begin{aligned}\nabla \cdot \sigma + f &= 0 \\ \tilde{\eta}_m - c \nabla^2 \tilde{\eta}_m &= \eta_m \\ \nabla \tilde{\eta}_m \cdot n_b &= 0\end{aligned}$$

The local variable η_m and the nonlocal variable $\tilde{\eta}_m$ are both composed of tension and compression parts:

$$\begin{aligned}\eta_m &= \begin{bmatrix} \eta_{mt} \\ \eta_{mc} \end{bmatrix} \\ \tilde{\eta}_m &= \begin{bmatrix} \tilde{\eta}_{mt} \\ \tilde{\eta}_{mc} \end{bmatrix}\end{aligned}$$

Each node, therefore, has two extra degrees of freedom.

There are 21 independent values of the equivalent strains η_t^{mic} and η_c^{mic} (one for each microplane). The tension and compression components of the local variable are evaluated by homogenizing the microplane values, as follows:

$$\begin{bmatrix} \eta_{mt} \\ \eta_{mc} \end{bmatrix} = \frac{1}{4\pi} \begin{bmatrix} \int_{\Omega} \eta_t^{mic} d\Omega \\ \int_{\Omega} \eta_c^{mic} d\Omega \end{bmatrix}$$

The regularization scheme is completed by the *over-nonlocal* formulation, where over-nonlocal variables $\hat{\eta}_t^{mic}$ and $\hat{\eta}_c^{mic}$ are evaluated as a linear combination of local and nonlocal variables, as follows:

$$\begin{aligned}\hat{\eta}_t^{mic} &= m \tilde{\eta}_{mt} + (1-m) \eta_t^{mic} \\ \hat{\eta}_c^{mic} &= m \tilde{\eta}_{mc} + (1-m) \eta_c^{mic}\end{aligned}$$

where m is a material parameter which should be > 1 to achieve regularization. Therefore, the nonlocal variable takes a weight larger than unity and the local variable is assigned a negative weight (hence the term *over-nonlocal*). The averaging ensures a smooth-deformation field, thereby preventing displacement discontinuities which can lead to an ill-posed boundary-value problem.

The regularized variable is used to calculate the damage driving variables γ_t^{mic} and γ_c^{mic} , as follows:

$$\gamma_t^{mic} = \begin{cases} \hat{\eta}_t^{mic} - \gamma_{t0} & \hat{\eta}_t^{mic} > \gamma_{t0} \\ 0 & \hat{\eta}_t^{mic} \leq \gamma_{t0} \end{cases}$$

$$\gamma_c^{mic} = \begin{cases} \hat{\eta}_c^{mic} - \gamma_{c0} & \hat{\eta}_c^{mic} > \gamma_{c0} \\ 0 & \hat{\eta}_c^{mic} \leq \gamma_{c0} \end{cases}$$

where γ_{t0} and γ_{c0} are the tension and compression damage thresholds, respectively.

For more information, see [Implicit Gradient Regularization in the Material Reference](#) (p. 110).

4.8.2.2.4. Coupled Damage-Plasticity Microplane Model Parameters

The coupled damage-plasticity microplane model requires 15 parameters:

Parameter Type	Parameter Subtype	Parameter	Description
Elasticity	--	E	Modulus of elasticity
	--	ν	Poisson's ratio
Plasticity	Drucker-Prager yield function	f_{uc}	Uniaxial compressive strength
		f_{bc}	Biaxial compressive strength
		f_{ut}	Uniaxial tensile strength
	Compression cap	σ_V^c	Intersection point abscissa between compression cap and Drucker-Prager yield function
		R	Ratio between the major and minor axes of the cap
	Hardening	D	Hardening material constant
		R_T	Tension cap hardening constant
Damage	--	γ_{t0}, γ_{c0}	Tension and compression damage thresholds
	--	β_t, β_c	Tension and compression damage evolution constants
Nonlocal	--	c	Nonlocal interaction range parameter
	--	m	Over-nonlocal averaging parameter

The plasticity parameters f_{uc} , f_{bc} , and f_{ut} are used as inputs because they are common material properties (or can be found experimentally) for materials such as concrete. The initial yield stress σ_0 and the friction coefficient α can be calculated by knowing that the biaxial and uniaxial stress states lie on the linear Drucker-Prager portion of the yield surface as follows:

$$\alpha = \frac{\sqrt{3}(f_{bc} - f_{uc})}{2f_{bc} - f_{uc}}$$

$$\sigma_0 = \left(1/\sqrt{3} - \alpha/3\right)f_{uc}$$

The following empirical relations are used to calculate the tension cap parameters: the abscissa of the intersection point between the tension cap and the Drucker-Prager yield function σ_V^T , and the initial intersection of the cap with the volumetric axis T_0 :

$$\sigma_V^T = -f_{uc}/3$$

$$T_0 = f_{ut}/3$$

4.8.2.2.5. Defining the Coupled Damage-Plasticity Microplane Model

The coupled damage-plasticity microplane model is used with the following coupled pore-pressure-thermal mechanical solid elements: [CPT212](#), [CPT213](#), [CPT215](#), [CPT216](#), and [CPT217](#).

To activate the required extra degrees of freedom (GFV1, GFV2), set KEYO(18) = 2. The extra degrees of freedom require no boundary condition input.

A fine mesh is recommended, particularly at probable damage-prone regions. To observe mesh-independent results, the mesh size may need to be less than half the square root of the nonlocal parameter c .

Define the elastic parameters E and ν via [TB,ELAS](#) or [MP](#).

Define the plasticity and damage parameters via [TB,MPLANE,,,DPC](#). Following are the material data table constants:

Constant	Meaning	Property	Range
C1	f_{uc}	Uniaxial compressive strength	$f_{uc} \geq f_{ut}$
C2	f_{bc}	Biaxial compressive strength	$f_{bc} \geq f_{uc}$
C3	f_{ut}	Uniaxial tensile strength	$f_{ut} \geq 0$
C4	R_T	Tension cap hardening constant	$R_T \geq 0$
C5	D	Hardening material constant	$D \geq 0$
C6	σ_V^C	Intersection point abscissa between compression cap and Drucker-Prager	$\sigma_V^C \leq -f_{uc}/3$

Constant	Meaning	Property	Range
		yield function	
C7	R	Ratio -- between the major and minor axes of the cap	$R \geq 0$
C8	γ_{t0}	Tension damage threshold	$\gamma_{t0} \geq 0$
C9	γ_{c0}	Compression damage threshold	$\gamma_{c0} \geq 0$
C10	β_t	Tension damage evolution constant	$\beta_t \geq 0$
C11	β_c	Compression damage evolution constant	$\beta_c \geq 0$

Define the nonlocal parameters via **TB,MPLANE,,,,,NLOCAL**. Following are the material data table constants:

Constant	Meaning	Property	Range
C1	c	Nonlocal interaction range parameter	$c \geq 0$
C2	m	Over-nonlocal averaging parameter	$m \geq 1$

Example 4.23: Coupled Damage-Plasticity Microplane Model Input

```

/prep7
!Define element
ET,1,215
KEYO,1,18,2 ! Activate extra degrees of freedom

! Parameter values
E = 28000
nu = .2
fuc = 30
fbc = 34.5
fut = 2.9

```

```

Rt = 1
D = 4e4
sigVc = -40
R = 2
c = 1500
m = 2.5
gamt0 = 0
gamc0 = 2e-6
betat = .4e4
betac = .25e4

! Define elastic properties of material
MP,EX,1,E
MP,NUXY,1,nu

! Define microplane model properties
TB,MPLA,1,,DPC
TB,DATA,1,fuc, fbc, fut, Rt, D, sigVc
TB,DATA,7,R, gamt0, gamc0, betat, betac

TB,MPLA,1,,NLOCAL
TB,DATA,1,c,m

```

To study a usage example for this material model, see [Reinforced Concrete Joint Analysis in the Technology Demonstration Guide](#).

4.8.2.2.6. Identifying Coupled Damage-Plasticity Microplane Model Parameters

Following are some hints and tips to help you with parameter identification.

- **Elasticity**

E and ν can be identified from the elastic region of the material stress-strain curve, or by using empirical formulas available in the literature.

- **Plasticity**

The strength parameters f_{uc} , f_{bc} , and f_{ut} are common material properties (or can be found experimentally) for materials such as concrete. In the absence of complete testing data, empirical relations [7] can be used if f_{uc} is known:

$$f_{bc} = 1.15 f_{uc}$$

$$f_{ut} = 1.4 \left(f_{uc} / 10 \right)^{2/3}$$

To identify the compression cap parameters, triaxial experimental data is necessary. The intersection point between the initial compression cap and the hydrostatic axis $X_0 + \sigma_V^C$ is determined by applying a hydrostatic load until the yielding begins.

The intersection point between the compression cap and the Drucker-Prager function σ_V^C is more challenging to find. It can be approximated as the transition point from plastic volumetric expansion (occurring on the linear Drucker-Prager function) to plastic volumetric compaction (occurring on the compression cap). If this data is unavailable, it can be estimated empirically as:

$$\sigma_V^C = -\frac{2}{3} f_{bc}$$

The parameter R (the ratio between the major and minor axes of the cap) can therefore be calculated as:

$$R = X_0 / f_1(\sigma_V^c)$$

- **Damage and Hardening**

To identify the damage and hardening parameters, cyclic tests are necessary. These parameters are related, as their interaction controls the softening and the unloading slope. A uniaxial cyclic compression test identifies the parameters D , β_c , and γ_{c0} .

Similarly, a uniaxial cyclic tension test identifies the parameters R_T , β_t , and γ_{t0} . In the absence of uniaxial cyclic tension tests, $R_T=1$ and $\beta_t=1.5\beta_c$ can be used as starting values. The tension damage threshold γ_{t0} is often set to zero, as softening in tension starts almost immediately after the elastic limit.

- **Nonlocal Parameters**

The over-nonlocal averaging parameter m is a numerical parameter, where $m > 1$ regularizes the solution. Typically, $m = 2.5$.

Also see [Identifying the Nonlocal Interaction-Range Parameter \$c\$](#) (p. 110).

4.8.2.2.7. Coupled Damage-Plasticity Microplane Damage Output

Certain microplane element quantities are available for postprocessing (**/POST1**, **/POST26**). The assigned label is MPDP.

Element outputs include the homogenized total damage (TOTA), homogenized tension damage (TENS), homogenized compression damage (COMP), and the split weight factor (RW).

4.8.2.3. Solution Control for the Regularized Microplane Damage Models

This section applies to both the [regularized elastic damage microplane model](#) (p. 113) and the [coupled damage-plasticity microplane model](#) (p. 116).

The Newton-Raphson out-of-balance loads caused by the extra degrees of freedom can be controlled by setting the reference value and tolerance for the gradient field residual (**CNVTOL**, **GFRS**).

Example 4.24: Setting the Reference Value and Tolerance for the Gradient Field Residual

```
CNVTOL, GFRS, 1e-7, .001
```

Automatic time-stepping uses an internal heuristic to adjust the time increment. You can set an additional time-stepping control (**CUTCONTROL**, **MDMG**) to limit the maximum allowable microplane homogenized damage increment in a time step.

Example 4.25: Setting a Time-stepping Control

```
CUTCONTROL, MDMG, 1e-3
```

4.8.3. Learning More About Microplane Material Modeling

The following list of resources offers more information about microplane material modeling:

1. Bazant, Z. P., P.G. Gambarova. "Crack Shear in Concrete: Crack Band Microplane Model." *Journal of Structural Engineering* . 110 (1984): 2015-2036.
2. Bazant, Z. P., B. H. Oh. "Microplane Model for Progressive Fracture of Concrete and Rock." *Journal for Engineering Mechanics* . 111 (1985): 559-582.
3. Leukart, M., E. Ramm. "A Comparison of Damage Models Formulated on Different Material Scales." *Computational Materials Science*. 28.3-4 (2003): 749-762.
4. Zreid, Imadeddin and Michael Kaliske. "A Gradient Enhanced Plasticity-Damage Microplane Model for Concrete." *Computational Mechanics*. 10.1007/s00466-018-1561-1 (2018).
5. Zreid, Imadeddin and Michael Kaliske. "An Implicit Gradient Formulation for Microplane Drucker-Prager Plasticity." *International Journal of Plasticity*. 83 (2016): 252-272.
6. Zreid, Imadeddin and Michael Kaliske. "Regularization of Microplane Damage Models Using an Implicit Gradient Enhancement." *International Journal of Solids and Structures*. 51.19 (2014): 3480-3489.
7. Jiang, H., J. Zhao. "Calibration of the Continuous Surface Cap Model for Concrete." *Finite Elements in Analysis and Design*. 97 (2015): 1-19.
8. Bažant, Z. P., G. Pijaudier-Cabot. "Measurement of Characteristic Length of Nonlocal Continuum." *Journal of Engineering Mechanics*. 115.4 (1989): 755-767.
9. Le Bellégo, C., J. F. Dubé, G. Pijaudier-Cabot, B. Gérard. "Calibration of Nonlocal Damage Model from Size Effect Tests." *European Journal of Mechanics-A/Solids*. 22.1 (2003): 33-46.
10. Xenos, D., D. Grégoire, S. Morel, P. Grassl. "Calibration of Nonlocal Models for Tensile Fracture in Quasi-brittle Heterogeneous Materials." *Journal of the Mechanics and Physics of Solids*. 82 (2015): 48-60.

4.9. Geomechanics

Geomechanics encompasses the mechanical behavior of soil, rock, and aggregate materials in both their natural and man-made states. Applications for these material models include footings and pilings, tunneling, excavations, seismic events, and compaction or consolidation. Other applications include material failure along stress concentrations or weak regions such as the failure of intact and jointed rock, masonry structures, and the crushing or failure behavior of concrete.

The material behavior is characterized by an initial elastic response followed by plastic deformation and unloading from the plastic state recovers the elastic deformation. The plasticity is a result of the microscopic behavior of the material particles and includes shear loading that causes particles to move past one another, changes in void or fluid content that result in volumetric plasticity, and exceeding the cohesive forces between the particles or aggregates.

The following topics about geomechanical plasticity theory, behavior and model definition are available:

[4.9.1. Understanding the Material Models for Geomechanics](#)

[4.9.2. Cam-clay](#)

[4.9.3. Mohr-Coulomb](#)

4.9.4. Jointed Rock

4.9.5. Drucker-Prager Concrete

4.9.6. Menetrey-Willam

Also see:

- Porous Elasticity in the *Material Reference* (p. 26)
- Initial State in the *Advanced Analysis Guide*
- Active and Passive Lateral Earth-Pressure Analysis in the *Technology Demonstration Guide*
- Load-Limit Analysis of a Reinforced Concrete Slab in the *Technology Demonstration Guide*
- Sequential Construction of an Embankment on a Clay Layer in the *Technology Demonstration Guide*

4.9.1. Understanding the Material Models for Geomechanics

Material behavior is defined by [rate-independent plasticity](#) (p. 28). The general concepts for the models are given in [Understanding the Plasticity Models](#) (p. 29).

Several of the yield surfaces for the geomechanics materials are defined by multiple surfaces. The elastic domain is the interior of the intersection of the surfaces, resulting in a continuous but non-smooth composite yield surface. Yielding and plastic deformation on one of the yield surfaces is the same as that of other rate-independent plasticity models. Plasticity and yielding can also occur at the intersection of two or more yield surfaces, resulting in plastic behavior that is a combination of the multiple yield surfaces [1].

When yielding occurs on multiple yield surfaces, the plastic strain increment is given by:

$$d\epsilon^{pl} = \sum_n d\lambda^n \frac{\partial Q^n}{\partial \sigma}$$

where $d\epsilon^{pl}$ is the increment in plastic strain, n ranges over the set of active yield surfaces, $d\lambda^n$ is the magnitude of the plastic strain increment, and Q^n is the flow potential.

The hardening or softening of the yield surfaces is a function of the hardening variable given by:

$$d\kappa = \sum_n d\lambda^n h^n$$

where $d\kappa$ is the increment in the hardening variable and h^n is the hardening modulus.

The incremental equations for rate-independent multisurface plasticity result in a system of nonlinear equations for the plastic strain increment, hardening variable increment and stress. One of two well-known integration methods, commonly called return mapping methods, is used to solve these equations: (1) the closest-point projection; and (2) the cutting-plane method. The details for these methods can be found in [2], but other considerations apply.

The closest-point projection is the more accurate method but generally has a smaller radius of convergence and can diverge when the stress state is near the intersection of two or more yield surfaces. This method returns a consistent material tangent, generally resulting in rapid convergence of the

Newton-Raphson iterations. For multisurface yielding, however, the consistent tangent can be incorrect when the activity of the yield surfaces changes over an increment; although this behavior has no effect on the accuracy of the converged solution, it can affect the convergence rate. The closest-point projection is used to solve all of the geomechanics models except for the [Mohr-Coulomb \(p. 136\)](#) and [jointed rock \(p. 145\)](#) models.

While the cutting-plane method tends to be more stable, it is less accurate, and does not have a consistent material tangent (instead using the material elastic tangent). The accuracy of the model can be checked heuristically by reducing the strain increments to determine if the solution changes. Use of an elastic material tangent reduces the rate of convergence of the Newton-Raphson iterations for the global system of equations, but does not affect the accuracy of the converged solution. The cutting-plane method is used to solve Mohr-Coulomb and jointed rock models.

4.9.1.1. Guidance for Using the Geomechanics Material Models

Simulations involving the geomechanics materials can sometimes present solution difficulties:

- **Cutting-plane algorithm**

The Mohr-Coulomb and jointed rock materials use the cutting-plane algorithm to solve the system of nonlinear equations that determine the material state at each element integration point. The algorithm tends to be more robust for these materials because of the non-smooth yield surface; however, it also results in an elastic material tangent, reducing the convergence rate of the Newton-Raphson iterations for the global system of equations. It may be necessary therefore to increase the maximum number of equilibrium iterations ([NEQIT](#)). Also, the elastic material tangent affects the results of any subsequent analyses that depend on the material stiffness (such as a perturbation analysis).

- **Material models that use the closest-point projection**

When the plastic flow direction is not associated with the stress direction, the material tangent is unsymmetric. This unassociated flow occurs when the plastic potential is different from the yield surface. For such cases, the convergence rate can be increased by using the unsymmetric solver ([NROPT](#)).

- **Plastic behavior and failure**

The plastic behavior for some of the geomechanical materials represents the sudden failure typical of geotechnical and aggregate materials. The behavior includes perfect plasticity and non-smooth or continuous softening, where further loading does not experience any resistance to deformation. The behavior can cause convergence difficulty for the global Newton-Raphson iterations.

The best method for overcoming the problem is to design a simulation that includes only localized yielding, with surrounding structure that prevents uncontrolled deformation of the areas of the mesh that have failed or softened. In some cases where the material has failed, stabilization or using a transient analysis can prevent uncontrolled deformation. In general, displacement boundary conditions that are applied or transferred through the mesh to material that has failed are more easily solved than similar force boundary conditions.

Material failure can often result in a localized area of high deformation (representing the total failure of the material). The localization is a smeared representation of a crack, shear band, or other failure, but the details of the localization often depend on the simulation setup (including

mesh size, substep size or loading rate, and material tangent formulation). While the predicted failure region might be accurate, the details of the localization are often unreliable.

4.9.1.2. Material Solution Option

Softening in the stress-strain response, often used to represent failure in geomechanics materials, can cause convergence difficulty for the global Newton-Raphson iterations. To help overcome convergence issues, you can use the material elastic tangent instead of the consistent material tangent.

A consistent tangent generally results in fewer Newton-Raphson iterations. When using an elastic tangent, it may be necessary to increase the maximum number of Newton-Raphson iterations (NEQIT).

For each of the geomechanics materials ([Cam-clay \(p. 131\)](#) soil, [Mohr-Coulomb \(p. 136\)](#), [jointed rock \(p. 145\)](#), [Drucker-Prager \(p. 153\)](#) concrete, and [Menetrey-Willam \(p. 167\)](#) concrete), a data table with the material solution option (*TBOPT* = MSOL) defines the form of the material tangent. The default value is used if the parameter is not redefined (or is defined as zero) in the material data table (**TB**).

Table 4.3: Material Solution Parameter

Constant	Property	Default Value	Range
C1	Elastoplastic tangent flag	Material-dependent	1 = Consistent tangent 2 = Elastic tangent

The default value for the Cam-clay, Drucker-Prager and Menetrey-Willam models returns the consistent algorithmic tangent. For the Mohr-Coulomb and jointed rock materials, the elastic tangent is the default material tangent.

Example 4.26: Drucker-Prager Concrete Material Using Elastic Tangent

```
/prep7
TB, CONCR, 1, , , MSOL
TBDATA, 1, 2
```

4.9.1.3. Nomenclature

Following are the common symbols used in the geomechanical theory documentation:

Symbol	Definition	Symbol	Definition
ε^p	Plastic strain	p_0	Initial pressure for porous elasticity
$d\lambda$	Magnitude of plastic strain increment	c	Cohesion
Q	Plastic potential	c'	Residual cohesion
κ	Hardening variable or swell index	ϕ	Friction angle
h	Hardening modulus	ϕ'	Residual friction angle
f	Yield function	ψ	Dilatancy angle
f'	Residual yield function	T	Tensile strength

Symbol	Definition	Symbol	Definition
p	Pressure	T'	Residual tensile strength
t	Modified stress invariant	R_{sc}	Residual strength coupling flag
σ	Stress	α_j, β_j	Failure plane orientation angles
σ_e	Von Mises effective stress	R_c	Compressive strength
J_2	2 nd principal invariant of the stress tensor	R_t	Tensile strength
J_3	3 rd principal invariant of the stress tensor	R_b	Biaxial compressive strength
τ	Effective shear stress	δ_t	Tension dilatancy parameter
σ_m	Mean stress	$\delta_{t1}, \delta_{t2}, \delta_{t3}, \delta_{t4}$	Tension dilatancy parameter points
σ_n	Normal stress	δ_c	Compression dilatancy parameter
a_h	Cam-clay yield surface hardening and softening variable	κ_{cm}	Plastic strain at uniaxial compressive strength
e_0	Initial porosity	κ_{cu}	Plastic strain at transition point
e	Porosity	κ_{cr}	Ultimate effective plastic strain
θ	Load angle	$\kappa_{t1}, \kappa_{t2}, \kappa_{t3}$	Softening plastic strain points
Ω_1	Hardening/softening function	Ω_{ci}	Relative stress at onset of nonlinear hardening
Ω_t	Tensile hardening/softening function	Ω_{cr}	Residual compressive relative stress
Ω_c	Compressive hardening/softening function	Ω_{tr}	Residual tensile relative stress
$\beta, \beta_{dry}, \beta_{wet}$	Yield surface shape parameter	Ω_{cu}	Residual relative stress at transition
a_{h0}	Initial yield surface size parameter	$\Omega_{t1}, \Omega_{t2}, \Omega_{t3}$	Residual tensile relative stress points
$a_{h,min}$	Minimum yield surface size parameter	κ_{tr}	Plastic strain limit in tension
M_c	Parameter for slope of the critical state line	G_{ft}	Mode I fracture energy in tension
K_s	Yield surface shape parameter	G_{fc}	Mode I fracture energy in compression
λ_F	Plastic slope parameter	--	--

4.9.1.4. Resources

The following list of resources offers more information about geomechanical plasticity:

1. Simo, J. C., J. G. Kennedy, S. Govindjee. "Non-smooth multisurface plasticity and viscoplasticity. Loading/unloading conditions and numerical algorithms." *International Journal for Numerical Methods in Engineering*. 26.10 (1988): 2161-2185.
2. Simo, J. C., T. J. R. Hughes. *Computational Inelasticity*. Springer, 1998.

3. Roscoe, K., J. B. Burland. "On the Generalised Stress-Strain Behaviour of Wet Clay." *Engineering Plasticity*. 169.1 (1968): 535-609.
4. Wood, D. M. *Soil Behaviour and Critical State Soil Mechanics*. Cambridge University Press, 1990.
5. Bazant, Z. P., B. H. Oh. "Crack Band Theory for Fracture of Concrete." *Materials and Structures*. 16.3 (1983): 155-177.

4.9.2. Cam-clay

The modified Cam-clay plasticity model [3] is based on the critical state concept and is commonly used for soil simulation. The critical state concept is a material phenomenon common in the shear deformation of soils where the stress and volume remain constant after a critical state of deformation, and further loading causes increasing plastic strain but no increase in volume or stress. The Cam-clay plasticity model, combined with [porous elasticity \(p. 26\)](#), models the effect of voids on the elastic behavior of the material.

The following topics for the Cam-clay model are available:

[4.9.2.1. Yield Surface](#)

[4.9.2.2. Defining the Cam-clay Model](#)

Also see [Sequential Construction of an Embankment on a Clay Layer in the Technology Demonstration Guide](#).

4.9.2.1. Yield Surface

The yield surface is a function of pressure p and a modified stress invariant t :

$$f_{cc} = \frac{1}{\beta^2} \left(\frac{p}{a_h} - 1 \right)^2 + \left(\frac{t}{M_c a_h} \right)^2 - 1 \quad (4.41)$$

where β , a_h , and M_c are material parameters. The modified Cam-clay model uses an associated flow rule, so the flow potential is the same as the yield function and plastic flow is normal to the yield surface.

Figure 4.28: Modified Cam-clay Yield Surface Plotted in Principal Stresses



Pressure is a function of elastic volumetric strain and void ratio and is defined by the [porous elasticity \(p. 26\)](#) model. The modified stress invariant is:

$$t = \frac{\sigma_e}{g}$$

where σ_e is the von Mises effective stress

$$\sigma_e = \sqrt{3J_2}$$

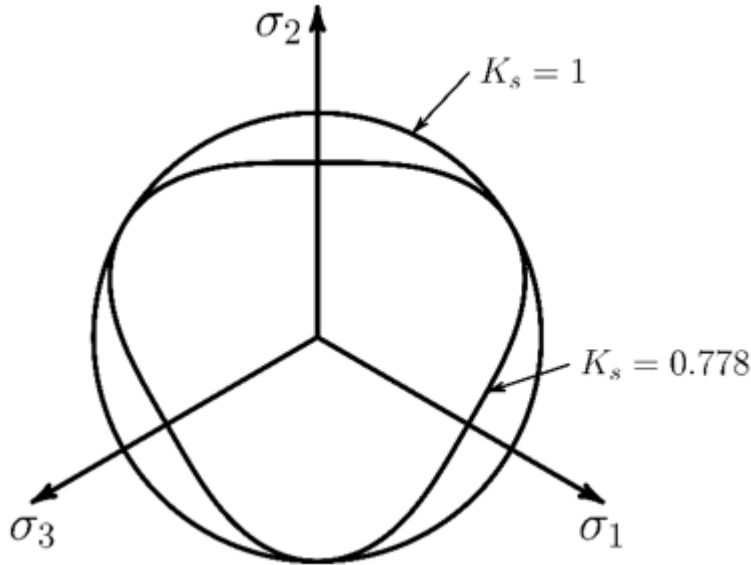
and

$$g = \frac{2K_s\sigma_e^3}{\sigma_e^3(1+K_s) + \frac{27}{2}(1-K_s)J_3}$$

where J_2 and J_3 are the second and third principal invariants of the stress tensor, respectively, and K_s is a material parameter that modifies the shape of the yield surface.

In the following figure, the yield surface is plotted in the octahedral plane (the intersection of the yield surface with a plane of constant pressure and plotted in principal stress space):

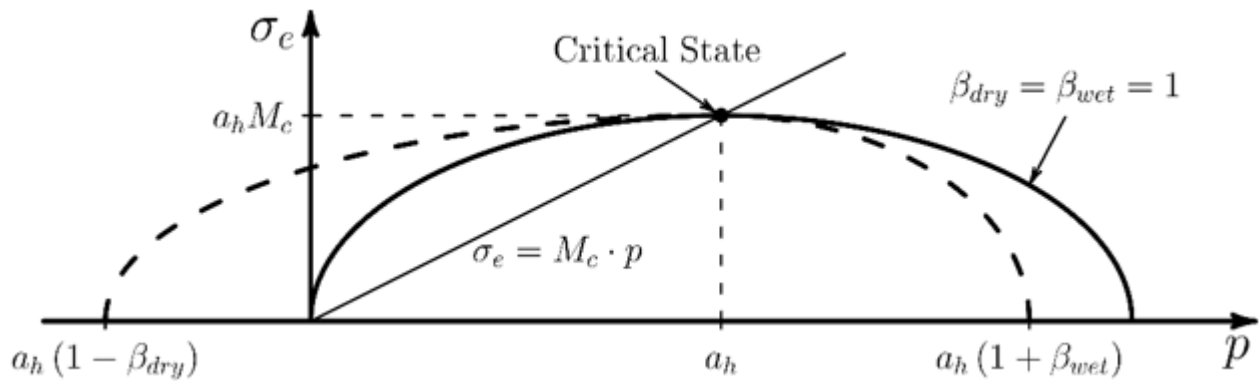
Figure 4.29: Modified Cam-clay Yield Surface in the Octahedral Plane



$K_s=1$ gives a circular yield surface and decreasing values cause the surface to become more triangular. To ensure that the yield surface is convex, $0.778 \leq K_s \leq 1.0$ is required.

In the following figure, the yield surface is plotted as effective stress versus pressure (q-p plane):

Figure 4.30: Modified Cam-clay Yield Surface in the q-p Plane



The critical state is then defined by the intersection of the yield surface with a line of slope M_c . The intersection, or *critical state*, is at the maximum value of effective stress.

Stress states above the critical state line are *over-consolidated*, and stress states below the line are *normally-consolidated*.

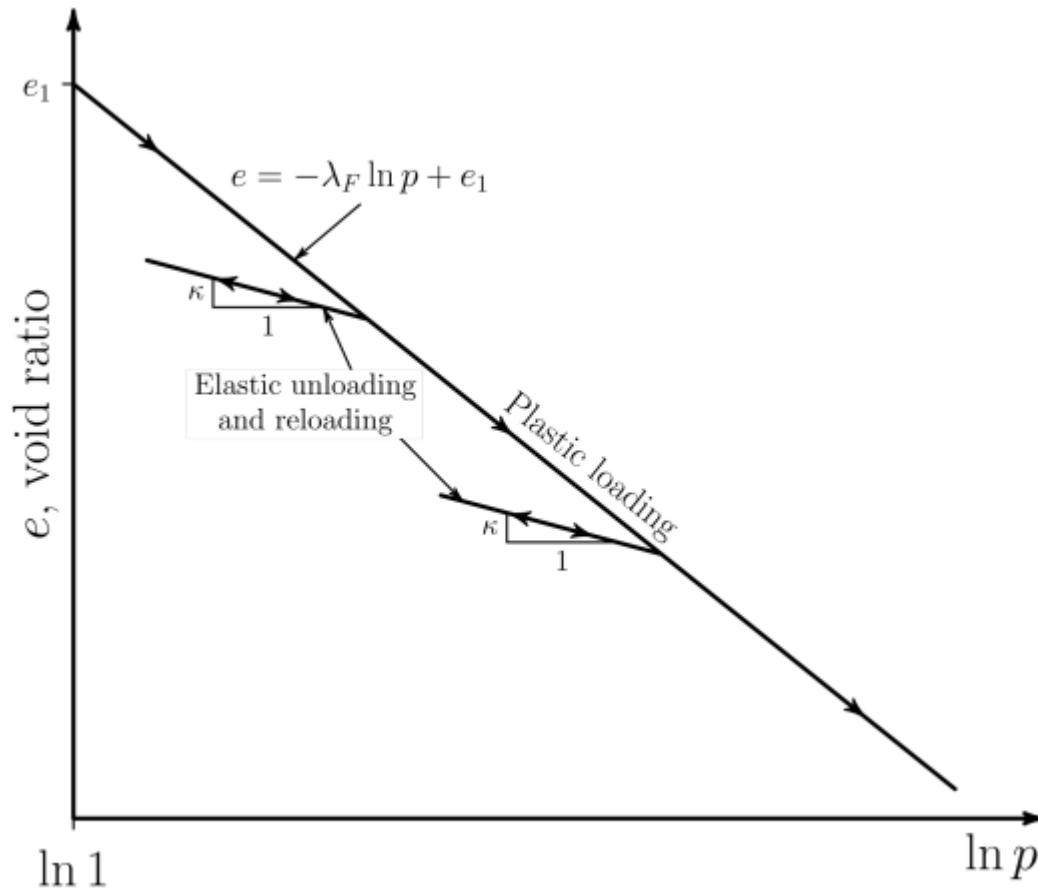
The plastic strain is normal to the yield surface so that yielding for over-consolidated states results in a decreasing plastic volumetric strain, and yielding for normally-consolidated states results in an increase in plastic volumetric strain. At the intersection of the critical state line with the yield surface, yielding causes no change in the plastic volume.

The part of the yield surface above the critical state line is called the *dry yield surface* and the yield surface below it is the *wet yield surface*. The parameter β modifies the shape of the yield surface, and separate values are used for the wet and dry surfaces. On the dry yield surface, β_{dry} is proportional to the cohesion. For no cohesion, $\beta_{dry}=1$ and the yield surface includes no tensile stress states. As β_{dry} increases, the elastic range for tensile stress states increases. For typical soil behavior, β_{dry} increases by only a few percent, whereas for high cohesion materials such as metal foams, β_{dry} can increase significantly.

Similar to a yield stress, a_h is a hardening or softening variable controlling the size of the yield surface. The initial size is defined by the parameter a_{h0} , and the hardening/softening behavior is given by:

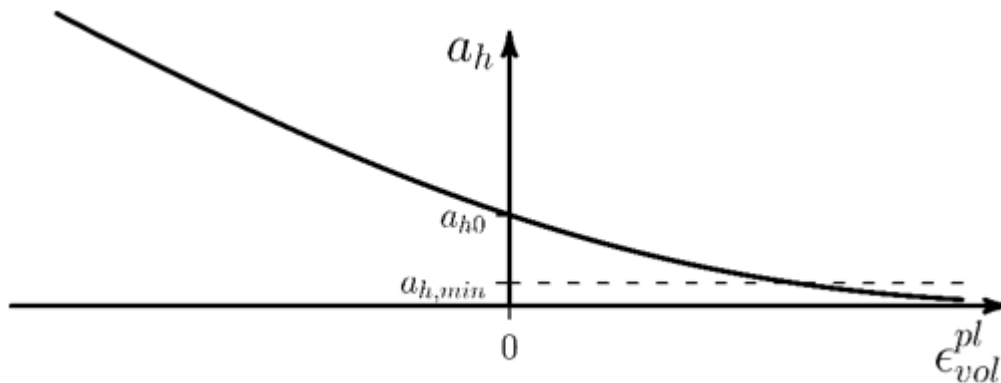
$$a_h = a_{h0} \exp \left((1 + e_0) \frac{1 - \exp(\epsilon_{vol}^{pl})}{\lambda_F - \kappa \exp(\epsilon_{vol}^{pl})} \right)$$

where e_0 is the initial porosity and κ is the swell index defined with the [porous elasticity \(p. 26\)](#) parameters. λ_F is a parameter for plastic slope from the assumed linear relationship between void ratio and natural log of the pressure during plastic deformation $de = -\lambda_F d(\ln p)$.

Figure 4.31: Linear Relationship Between Void Ratio and Natural Log of Pressure

Because κ is the slope of this line during elastic loading, λ_F must be larger than κ and is typically 3 to 6 times the value of κ .

The following figure shows the behavior of a_{h0} , where hardening occurs when the material is consolidating due to yielding above the critical state, and softening occurs when the materials is dilating due to yielding below the critical state:

Figure 4.32: Hardening and Softening Behavior

When defining an [initial stress](#) state, the stress should be inside or on the yield surface. If the initial stress is outside the yield surface, a_{h0} is adjusted so that the initial stress state is on the yield surface.

Softening of a_h can reduce the size of the yield surface and cause numerical difficulties. To prevent the yield surface from becoming too small, the minimum size is set via the parameter $a_{h,min}$.

4.9.2.2. Defining the Cam-clay Model

To use the Cam-clay model:

1. Define the porous elasticity (p. 26) material model (**TB,PELAS,,,,POISSON**) and required constants (p. 28).
2. Define the Cam-clay plasticity parameters (**TB,SOIL,,,,CAMCLAY**).
3. Specify the required Cam-clay material model constants (**TB,DATA**).
4. Define the initial stress state (p_0) (**INISTATE**).

Table 4.4: Cam-clay Model Constants

Constant	Meaning	Property	Unit	Range
C1	λ_F	Plastic slope parameter	--	$\lambda_F > \kappa > 0$
C2	M_c	Slope of critical state line	--	$M_c > 0$
C3	a_{h0}	Initial yield surface size	Force/Length ²	$a_{h0} > 0$
C4	$a_{h,min}$	Minimum yield surface size	Force/Length ²	$0 < a_{h,min} \leq a_{h0}$
C5	β_{dry}	Dry part of yield surface modifier	--	$\beta_{dry} \geq 1$
C6	β_{wet}	Wet part of yield surface modifier	--	$0 < \beta_{wet} \leq 1$
C7	K_s	Anisotropic yield surface parameter	--	$0.778 \leq K_s \leq 1$

Define temperature- or field-dependent data for the data tables via the **TBTEMP** or **TBFIELD** commands, respectively.

Example 4.27: Defining the Cam-clay Model

```

/prep7

! Porous elasticity
Kappa = 0.0024
NU0    = 0.279
pt_el  = 5.835
E0     = 0.34
p0     = 69

TB,PELAS,1,,,POISSON
TB,DATA,1, Kappa, pt_el, NU0, E0

! Cam-clay Plasticity
Lambda_F = 0.014
Mc       = 1.24
a0       = 35
ah_min   = 0.35
Beta_Dry = 1
Beta_Wet = 1
Ks       = 1

TB,SOIL,1,,,CAMCLAY
TB,DATA,1, Lambda_F, Mc, a0, ah_min
TB,DATA,5, Beta_Dry, Beta_Wet, Ks

```

```
/solu
!define initial stress state
INISTATE,set,dtyp,stre
INISTATE,defi,all,,, -p0, -p0, -p0, 0, 0, 0
```

4.9.3. Mohr-Coulomb

Aggregate materials such as soil, rock and concrete begin to plastically deform when the shear stress exceeds the internal friction resistance between the material particles. The friction resistance is a function of the normal force between the particles. Use the Mohr-Coulomb material model to represent such aggregate materials.

The following topics about the Mohr-Coulomb model are available:

[4.9.3.1. Understanding the Mohr-Coulomb Model](#)

[4.9.3.2. Defining the Mohr-Coulomb Model](#)

Also see [Active and Passive Lateral Earth-Pressure Analysis](#) and [Suction-Pile Analysis](#) in the Technology Demonstration Guide.

4.9.3.1. Understanding the Mohr-Coulomb Model

The following topics are available:

[4.9.3.1.1. Yield Surface](#)

[4.9.3.1.2. Plastic Flow Potentials](#)

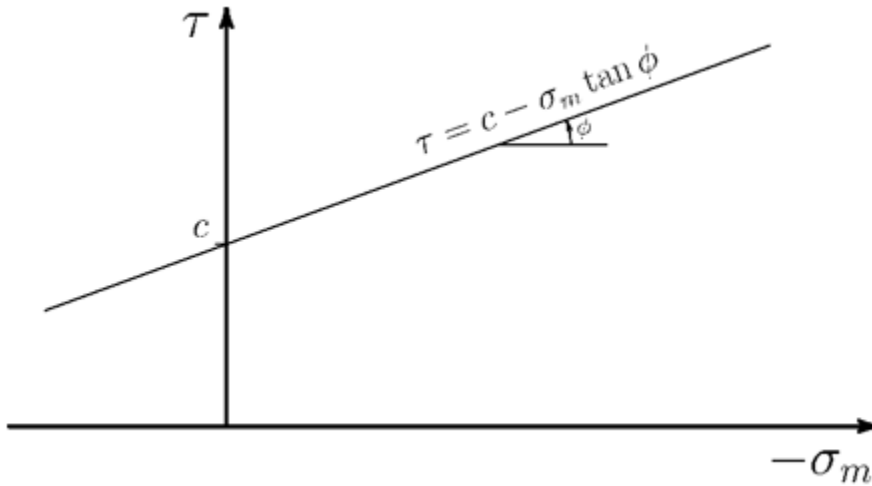
4.9.3.1.1. Yield Surface

The model defines yielding when the combination of pressure and shear stress reaches the cohesion of the material particles. Yielding occurs when the shear stress on any plane in the material reaches this criterion:

$$\tau = c - \sigma_m \tan \phi$$

where τ is the shear stress, c is the cohesion, σ_m is the mean stress, and ϕ is the inner friction angle.

In the following figure, the cohesion is the shear stress that causes yielding when the mean stress is zero and the friction angle defines the increase in the yield condition as the mean stress increases:

Figure 4.33: Mohr-Coulomb Yield Surface as Shear vs. Mean Stress

The friction angle is proportional to the stress required to shear particles past one another. Materials such as loose sand, in which the particles can easily move past one another, have relatively low friction angle.

Generalization to any state of stress gives the Mohr-Coulomb yield surface:

$$f_{MC}(\sigma) = \sigma_m \sin \phi + \frac{\sigma_e}{\sqrt{3}} \left(\cos \theta - \frac{\sin \theta \sin \phi}{\sqrt{3}} \right) - c \cos \phi$$

where:

$$\sigma_m = \frac{\sigma_{11} + \sigma_{22} + \sigma_{33}}{3}$$

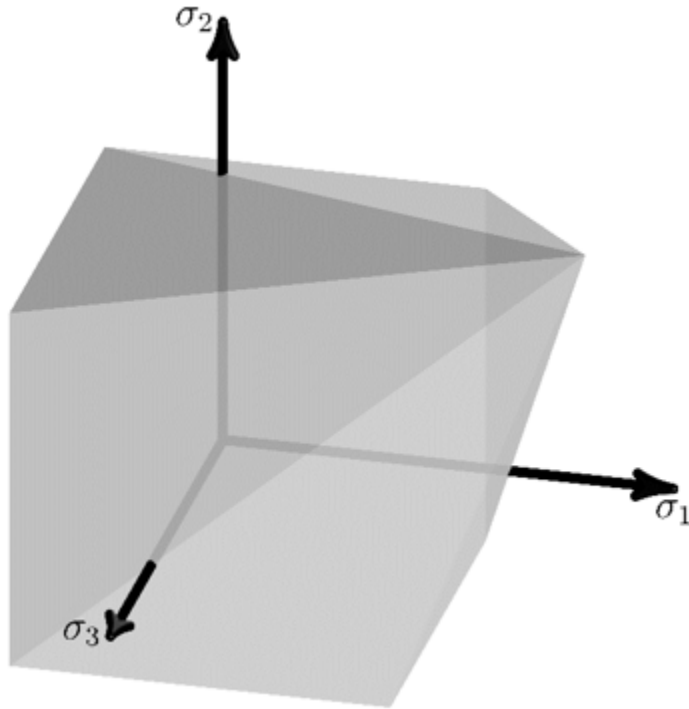
$$\sigma_e = \sqrt{3J_2}$$

$$\sin(3\theta) = -\frac{3\sqrt{3}}{2} \frac{J_3}{\sqrt{J_2^3}}$$

and the stress invariants are:

$$J_2 = \frac{1}{6} \left((\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 \right) + \sigma_{12}^2 + \sigma_{23}^2 + \sigma_{13}^2$$

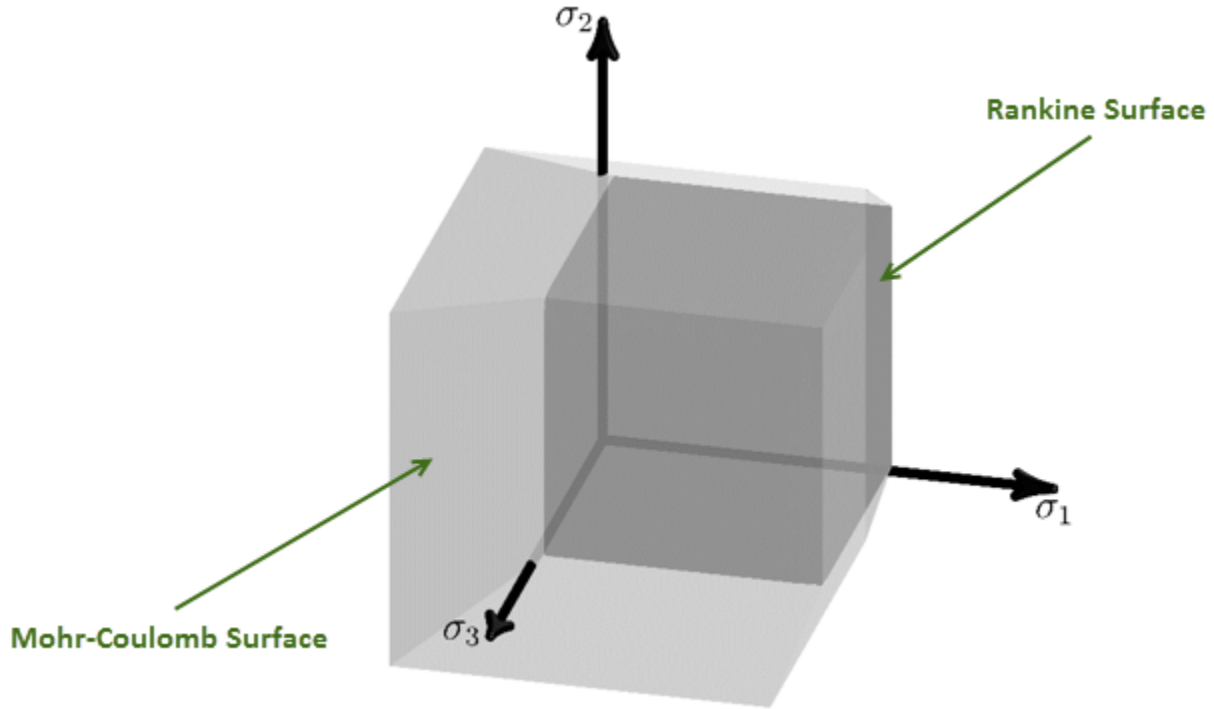
$$J_3 = \det(\sigma - I\sigma_m)$$

Figure 4.34: Mohr-Coulomb Yield Surface in Principal Stresses

The Mohr-Coulomb yield surface can be combined with a tension-failure surface to limit the material strength in tension. The tension yield surface is defined by a Rankine yield surface:

$$f_R(\sigma) = \sigma_m + \frac{2}{3} \sigma_e \sin(\theta + 120^\circ) - T$$

where T is the tensile strength.

Figure 4.35: Mohr-Coulomb with Tension-Failure Yield Surface in Principal Stress

After the material yields, the inner friction angle ϕ , cohesion c , and tensile strength T harden or soften as a function of the Mohr-Coulomb and tension hardening variables, κ_{MC} and κ_r . These hardening variables evolve by summing the incremental values for each substep and the incremental values are given by:

$$\Delta\kappa_{MC} = \sqrt{\frac{1}{6} \left[(\Delta\varepsilon_{xx}^{pl} - \Delta\varepsilon_{yy}^{pl})^2 + (\Delta\varepsilon_{yy}^{pl} - \Delta\varepsilon_{zz}^{pl})^2 + (\Delta\varepsilon_{zz}^{pl} - \Delta\varepsilon_{xx}^{pl})^2 \right] + \left(\frac{\Delta\varepsilon_{xy}^{pl}}{2} \right)^2 + \left(\frac{\Delta\varepsilon_{yz}^{pl}}{2} \right)^2 + \left(\frac{\Delta\varepsilon_{xz}^{pl}}{2} \right)^2}$$

$$\Delta\kappa_R = \frac{1}{T(\kappa_R)} \sigma \cdot \Delta\varepsilon^{pl}$$

The material parameters are then given by:

$$\begin{aligned}\phi(\kappa_{MC}) &= \Omega_\phi(\kappa_{MC})\phi \\ c(\kappa_{MC}) &= \Omega_c(\kappa_{MC})c \\ T(\kappa_R) &= \Omega_T(\kappa_R)T\end{aligned}$$

where ϕ , c , and T are the initial values and $\Omega_\phi(\kappa_{MC})$, $\Omega_c(\kappa_{MC})$, and $\Omega_T(\kappa_R)$ are user-defined hardening/softening scaling functions.

At zero plastic deformation, the scaling functions are equal to 1.0. It is good practice to define the value $\Omega(0)=1$ in the input so that the scaling functions are interpolated as expected for small values of plastic deformation. If any scaling function is defined, the undefined scaling functions are assumed constant $\Omega=1$.

If no scaling functions for the yield surface parameters are defined, then after initial yield, the yield surfaces reduce to the residual yield surfaces. If any parameter-scaling functions are defined, the residual parameters are ignored.

The residual Mohr-Coulomb yield surface is:

$$f'_{MC}(\sigma) = \sigma_m \sin \phi' + \frac{\sigma_e}{\sqrt{3}} \left(\cos \theta - \frac{\sin \theta \sin \phi'}{\sqrt{3}} \right) - c' \cos \phi'$$

where c' is the residual cohesion and ϕ' is the residual inner friction angle. To ensure a decrease in the yield strength, the residual yield surface constants should satisfy:

$$\frac{c}{\tan \phi} < \frac{c'}{\tan \phi'}$$

Otherwise, the residual cohesion is redefined as:

$$c' = \frac{c}{\tan \phi} \tan \phi'$$

The residual Rankine yield surface is:

$$f'_R(\sigma) = \sigma_m + \frac{2}{3} \sigma_e \sin(\theta + 120^\circ) - T'$$

Tension cutoff is defined at the intersection of the Mohr-Coulomb and Rankine yield surfaces:

$$I_1^* = \frac{c \cdot \cos \phi - T}{\sin \phi - 1}$$

$$\sqrt{J_2^*} = I_1^* - T$$

When the tension cutoff is defined, the residual Mohr-Coulomb yield surface parameters should satisfy:

$$c' \cdot \cos \phi' - I_1^* \cdot \sin \phi' > \sqrt{J_2^*}$$

Otherwise, the residual cohesion is redefined as:

$$c' = \frac{\sqrt{J_2^*} + I_1^* \cdot \sin \phi'}{\cos \phi'}$$

The residual strength of the Mohr-Coulomb surface and the tension-failure surfaces can be coupled so that when yielding occurs on one surface, the other yield surface is reduced to the residual strength yield function. The value of the flag defines the coupling:

$R_{SC}=0 \rightarrow$ No coupling between yield surface failures

$R_{SC}=2 \rightarrow$ Yielding on one surface reduces other surface to residual values

4.9.3.1.2. Plastic Flow Potentials

Three plastic flow potentials are available for the Mohr-Coulomb material model.

The Mohr-Coulomb flow potential is:

$$Q_{MCt} = \sigma_m \sin \psi + \frac{\sigma_e}{\sqrt{3}} \left(\cos \theta - \frac{\sin \theta \sin \psi}{\sqrt{3}} \right)$$

where ψ is the dilatancy angle. This is the only flow potential available when no scaling functions are defined for the yield surface parameters.

The Mohr-Coulomb potential is not smooth, requiring the material solution algorithm to use the cutting-plane return algorithm.

A smooth potential is obtained from the Mohr-Coulomb potential by fixing the value of θ :

- Setting $\theta = 30^\circ$ gives a potential surface that intersects the Mohr-Coulomb surface at the tension meridians (intersection of the Mohr-Coulomb surface with the positive principal stress axes):

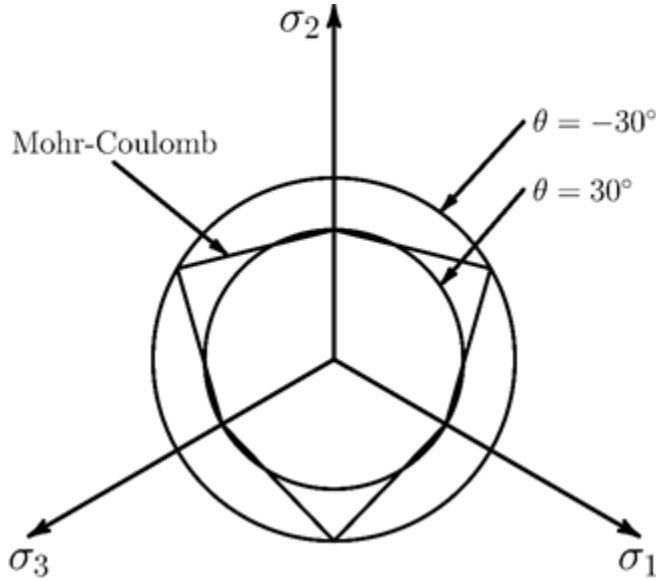
$$Q_{MC1} = \sigma_m \sin \psi + \frac{\sigma_e}{\sqrt{3}} \left(\frac{\sqrt{3}}{2} - \frac{1}{2} \frac{\sin \psi}{\sqrt{3}} \right)$$

- Setting $\theta = -30^\circ$ gives a potential surface that intersects the Mohr-Coulomb surface at the compression meridians (the intersection of the Mohr-Coulomb surface with the negative principal stress axes):

$$Q_{MC2} = \sigma_m \sin \psi + \frac{\sigma_e}{\sqrt{3}} \left(\frac{\sqrt{3}}{2} + \frac{1}{2} \frac{\sin \psi}{\sqrt{3}} \right)$$

A smooth potential enables the use of the closest-point projection method.

Using a smooth potential requires the material [scaling parameters](#). If none are defined, the parameters are constant.

Figure 4.36: Flow Potentials in the Octahedral Plane

Similar to the yield surface parameters, the dilatancy angle can be defined to evolve based on the scaling:

$$\psi(\kappa_{MC}) = \Omega_{\psi}(\kappa_{MC})\psi$$

where ψ is the initial value of the dilatancy angle and $\Omega_{\psi}(\kappa_{MC})$ is the dilatancy angle scaling function.

When the Mohr-Coulomb material includes the Rankine tension failure surface, two plastic flow potentials for yielding on this surface are available. The potential is selected based on the flow potential used for the Mohr-Coulomb yield surface.

When the Mohr-Coulomb potential is used with the Mohr-Coulomb surface, the tension failure potential is the Rankine tension failure surface.

When one of the smooth potentials is used with the Mohr-Coulomb surface, the plastic flow potential for the tension failure surface is:

$$Q_{MCt} = \begin{cases} \sigma_1^2 & \sigma_2 \leq 0, \sigma_3 \leq 0, \\ \sigma_1^2 + \sigma_2^2 & \sigma_2 > 0, \sigma_3 \leq 0 \\ \sigma_1^2 + \sigma_2^2 + \sigma_3^2 & \sigma_2 > 0, \sigma_3 > 0 \end{cases}$$

where $\sigma_1 \geq \sigma_2 \geq \sigma_3$ are the ordered principal stresses.

4.9.3.2. Defining the Mohr-Coulomb Model

To use the Mohr-Coulomb model:

1. Define the isotropic or anisotropic elastic behavior (via **MP** commands or via the elastic material data table [**TB**,ELASTIC]).
2. Define the Mohr-Coulomb yield surface and flow potential (**TB**,MC,,,,BASE).

3. Input the model constants (**TB,DATA**).

Table 4.5: Mohr-Coulomb Model Constants

Constant	Meaning	Property	Unit	Range
C1	ϕ	Initial inner friction angle	Degrees	$0 \leq \phi < 90$
C2	c	Initial cohesion	Force/Length ²	$c \geq 0$
C3	ψ	Dilatancy angle	Degrees	$0 \leq \psi \leq \phi$
C4	ϕ'	Residual inner friction angle	Degrees	$0 \leq \phi' \leq \phi$
C5	c'	Residual cohesion	Force/Length ²	$0 \leq c' \leq c$

To define the optional tension Rankine yield surface, define the material data table (**TB,MC,,,,RCUT**) and input the following constants (**TB,DATA**):

Table 4.6: Tension Rankine Yield Surface Constants (Optional)

Constant	Meaning	Property	Unit	Range
C1	T	Initial tensile strength	Force/Length ²	$T > 0$
C2	T'	Residual tensile strength	Force/Length ²	$0 < T' \leq T$

To define the optional residual strength coupling, define the material data table (**TB,MC,,,,RSC**) and input the following constant (**TB,DATA**):

Table 4.7: Residual Strength Coupling Constants (Optional)

Constant	Meaning	Property	Unit	Range
C1	R_{SC}	Residual strength coupling flag	--	$R_{SC} = 0, 2$

To define the optional flow potential, define the material data table (**TB,MC,,,,POTN**), then define the flow type (**TB,DATA**):

Table 4.8: Plastic Potential (Optional)

Constant	Meaning	Property	Unit	Range
C1	–	Plastic potential <i>Type</i>	–	0, 1, 2
<i>Type</i> = 0 – Mohr-Coulomb potential surface (default). <i>Type</i> = 1 – Smoothed compression-meridian potential surface. <i>Type</i> = 2 – Smoothed tension-meridian potential surface.				

To define the optional scaling function for the inner friction angle, define the material data table (**TB,MC,,,,FRICTION**) and define the $(\kappa_{MC}, \Omega_\phi)$ points (**TBPT**):

Table 4.9: Friction Angle Scaling Function (Optional)

Constant	Meaning	Property	Unit	Range
X	κ_{MC}	Hardening variable	--	$\kappa_{MC} \geq 0$
Y	Ω_ϕ	Scaling function	--	$\Omega_\phi > 0$

To define the optional scaling function for the cohesion, define the material data table (TB,MC,,,,COHESION) and define the (κ_{MC}, Ω_c) points (TBPT):

Table 4.10: Cohesion Scaling Function (Optional)

Constant	Meaning	Property	Unit	Range
X	κ_{MC}	Hardening variable	--	$\kappa_{MC} \geq 0$
Y	Ω_c	Scaling function	--	$\Omega_c > 0$

To define the optional scaling function for the tension strength, define the material data table (TB,MC,,,,TENSION) and define the (κ_R, Ω_T) points (TBPT):

Table 4.11: Tension Strength Scaling Function (Optional)

Constant	Meaning	Property	Unit	Range
X	κ_R	Hardening variable	--	$\kappa_R \geq 0$
Y	Ω_T	Scaling function	--	$\Omega_T > 0$

To define the optional scaling function for the dilatation angle, define the material data table (TB,MC,,,,DILATATION) and define the $(\kappa_{MC}, \Omega_\psi)$ points (TBPT).

Table 4.12: Dilatation Angle Scaling Function (Optional)

Constant	Meaning	Property	Unit	Range
X	κ_{MC}	Hardening variable	--	$\kappa_{MC} \geq 0$
Y	Ω_ψ	Scaling function	--	$\Omega_\psi > 0$

Define temperature- or field-dependent data for the data tables via the **TBTEMP** or **TBFIELD** commands, respectively.

Example 4.28: Defining the Mohr-Coulomb Model

```

/prep7
! ELASTIC CONSTANTS
MP,EX,1,20.0E9
MP,NUXY,1,0.2

! Mohr Coulomb yield surface

phi=30      ! friction angle
c=8.7e6     ! cohesion
psi=15      ! dilatancy angle
phir=20     ! residual friction angle
cr=0.8*c    ! residual cohesion

sigt=3e6    ! tensile strength
sigtr=0.8*sigt ! residual tensile strength

TB,MC,1,,,BASE
TBDATA,1,phi,c,psi,phir,cr

TB,MC,1,,,RCUT
TBDATA,1,sigt,sigtr

TB,MC,1,,,RSC

```

```
TBDATA,1,0
```

Example 4.29: Defining the Mohr-Coulomb Model with Softening

```
/prep7
YoungsModulus= 20.0E9
NU      =0.2

mp,ex,1,YoungsModulus
mp,prxy,1,NU

! nonlinear parameters for intact rock
phig=30      ! friction angle
cg=8.7e6     ! cohesion
psig=15     ! dilatancy angle
sigt=3e6     ! tension failure

TB,MC,1,,,BASE
TBDATA,1,phig,cg,psig

TB,MC,1,,,RCUT
TBDATA,1,sigt

! plastic potential
TB,MC,1,,,POTN
TBDATA,1,1

! softening functions
TB,MC,1,,,friction
TBPT,, 0.0 ,1
TBPT,, 1E-4,35/30

TB,MC,1,,,dilatation
TBPT,, 0.0 ,1
TBPT,, 1E-4,0.9

TB,MC,1,,,cohesion
TBPT,, 0.0 ,1
TBPT,, 1E-4,0.95
TBPT,, 1E-3,0.75
```

4.9.4. Jointed Rock

Geologic and aggregate materials can have inhomogeneous behavior that causes weakness, represented as joints, along planes within the material. Examples of such planes are stratification planes, and geometric stress concentrations or regions of strain localization.

The behavior of the planes is defined by an anisotropic Mohr-Coulomb yield surface that depends on the normal and shear stress on the face of the planes. The planes are embedded in a base material that can be modeled as either elastic, or elastic-plastic with an isotropic Mohr-Coulomb yield surface.

The following topics about the jointed rock material model are available:

[4.9.4.1. Mohr-Coulomb Base Material](#)

[4.9.4.2. Anisotropic Mohr-Coulomb Joints](#)

[4.9.4.3. Defining the Jointed Rock Model](#)

4.9.4.1. Mohr-Coulomb Base Material

The base material for the jointed rock model can be either elastic, or elastic-plastic defined via the [Mohr-Coulomb model \(p. 136\)](#). The Mohr-Coulomb base material includes the Mohr-Coulomb yield surface and a tension-cutoff yield surface.

If the Mohr-Coulomb base material is not defined, the program considers the base material to be elastic, and the only plastic behavior occurs in the joints.

4.9.4.2. Anisotropic Mohr-Coulomb Joints

Failure of the joints is defined via an anisotropic Mohr-Coulomb failure model that can be combined with tensile failure. The anisotropic Mohr-Coulomb yield function defines the yield surface on a failure plane in terms of the resolved normal and shear stress on the plane:

$$f_j = \tau_r - \sigma_n \tan \phi_j - c_j$$

where τ_R is the magnitude of the resolved shear stress on the failure plane, σ_n is the normal stress on the failure plane, ϕ_j is the failure plane friction angle, and c_j is the failure plane cohesion.

After the failure plane yields, the residual yield surface of the failure plane is given by:

$$f'_j = \tau_r - \sigma_n \tan \phi'_j - c'_j$$

The flow potential for the failure plane is:

$$Q_j = \tau_r - \sigma_n \tan \psi_j$$

A tension cutoff can be defined to limit the normal stress on the plane. The tension-cutoff yield surface is:

$$f_{Tj} = \sigma_n - T_j$$

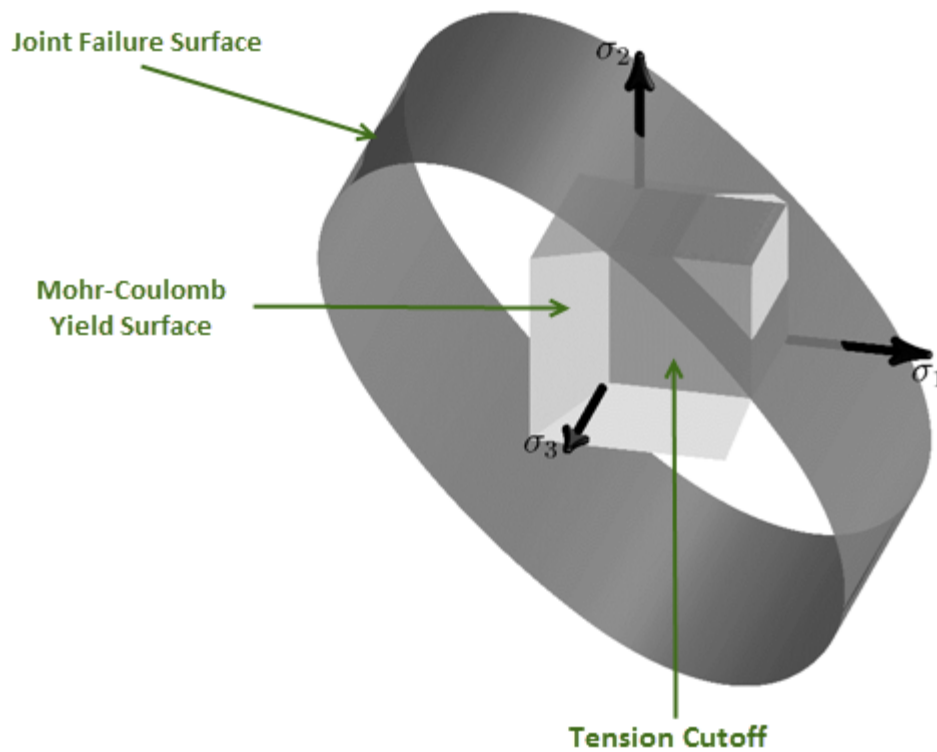
where T_j is the tensile strength of the failure plane.

The residual tensile strength yield surface is:

$$f'_{Tj} = \sigma_n - T'_j$$

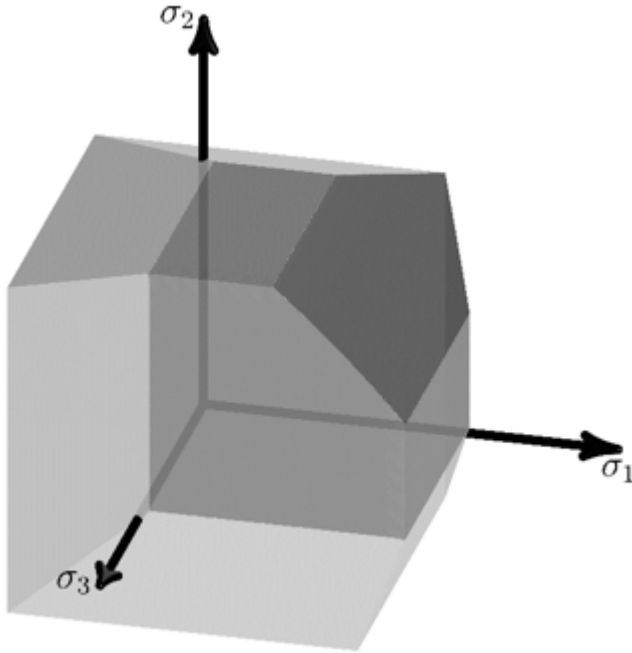
The following figure shows the anisotropic Mohr-Coulomb yield surface for a plane with normal $\vec{w} = (1, 1, 0)$:

Figure 4.37: Joint Failure Surface Superimposed on Mohr-Coulomb with Tension-Cutoff Yield Surface



The failure plane yield surface is superimposed on the base material yield surface, a Mohr-Coulomb model with a tension cutoff.

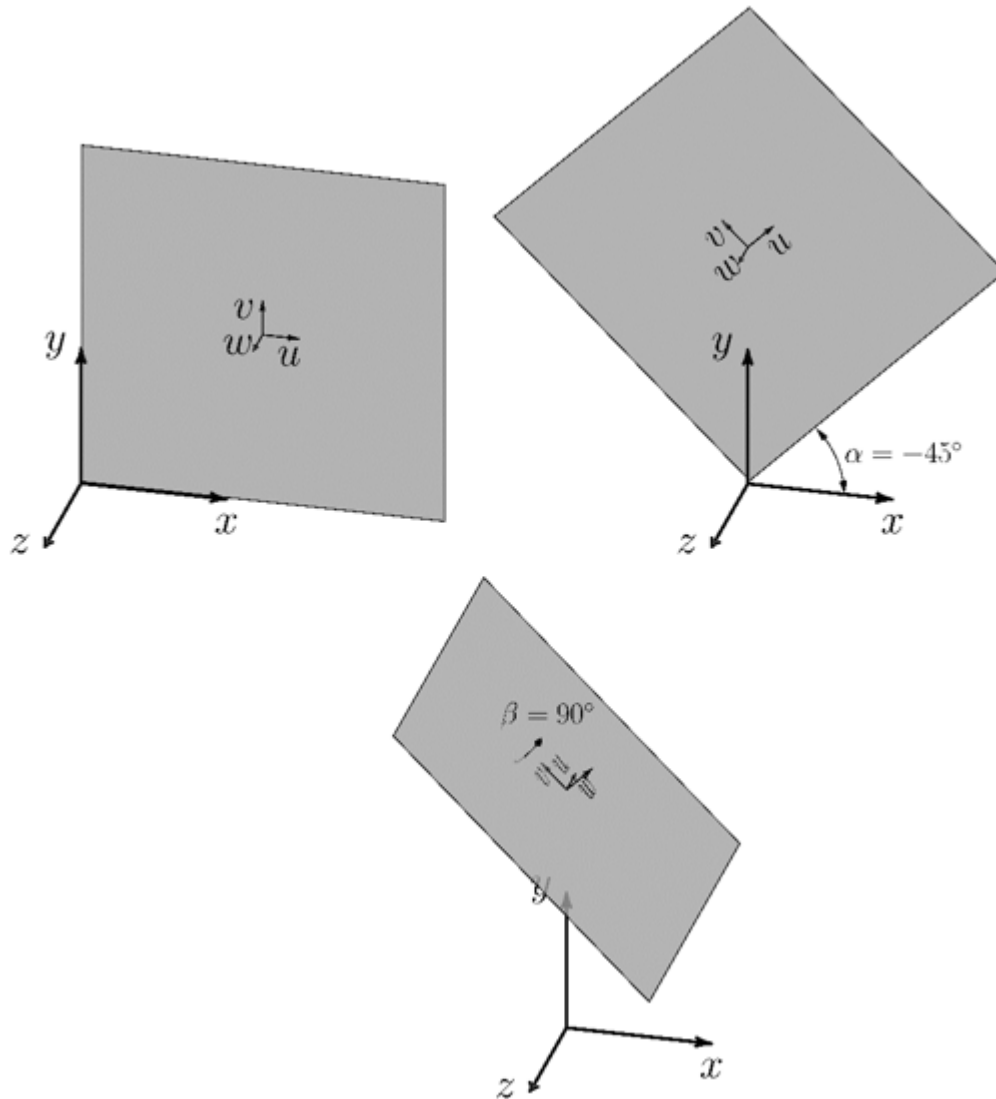
The following figure shows the composite yield surface in the tension region of the stress space in which the failure crops off a portion of the yield surface of the base material:

Figure 4.38: Composite Yield Surface of Jointed Rock with Mohr-Coulomb Base Material

The joints are oriented relative to the element coordinate system. By default, the plane has a normal in the z direction. The plane can be reoriented by defining angles α_j and β_j , where the plane is first rotated about the negative z axis by angle α_j and then rotated about the plane's y axis by angle β_j . The orientation of the failure plane normal $\vec{w}_j$ is:

$$\vec{w}_j = \begin{bmatrix} \cos(\alpha_j) \sin(\beta_j) \\ -\sin(\alpha_j) \sin(\beta_j) \\ \cos(\beta_j) \end{bmatrix}$$

The following figure shows an example of how a failure plane is reoriented:

Figure 4.39: Reorienting the Failure Plane

The plane has a local coordinate system $(\vec{u}, \vec{v}, \vec{w})$ initially aligned with the element coordinate system. It is reoriented by $\alpha = -45^\circ$ rotation about the z axis, then $\beta = 90^\circ$ rotation about the plane v axis.

The failure surfaces of the base material and the joints can be coupled by defining the residual strength coupling flag R_{SC} , as follows:

$R_{SC}=0 \rightarrow$ No coupling between yield surface failures

$R_{SC}=1 \rightarrow$ Yielding on one joint reduces other joint surfaces to residual values

$R_{SC}=2 \rightarrow$ Yielding on one base surface reduces other base surface to residual values

$R_{SC}=3 \rightarrow$ Yielding on either base or joint surface reduces all surfaces to residual values

4.9.4.3. Defining the Jointed Rock Model

Defining the jointed rock model consists of two parts:

4.9.4.3.1. Part 1: Defining the Base Material

4.9.4.3.2. Part 2: Defining the Joints

4.9.4.3.1. Part 1: Defining the Base Material

To use the jointed rock material model with an elastic base:

1. Define the isotropic or anisotropic elastic behavior (via **MP** commands or via the elastic material data table [**TB**,ELASTIC]).
2. Proceed to [Part 2: Defining the Joints \(p. 151\)](#).

In this case, the base model uses the elastic constants (from **MP** or **TB**,ELASTIC), and the only plastic behavior occurs in the joints.

To use the jointed rock material model with an elastic-plastic (Mohr-Coulomb) base:

1. Define the isotropic or anisotropic elastic behavior (via **MP** commands or via the elastic material data table [**TB**,ELASTIC]).
2. Define the jointed rock material data table (**TB**,JROCK,,,,BASE).
3. Input the required base material constants (**TB**DATA).

Table 4.13: Jointed Rock Constants for the Base Material

Constant	Meaning	Property	Unit	Range
C1	ϕ	Initial inner friction angle for base material	Degrees	$0 \leq \phi < 90$
C2	c	Initial cohesion for base material	Force/Length ²	$c \geq 0$
C3	ψ	Dilatancy angle for base material	Degrees	$0 \leq \psi \leq \phi$
C4	ϕ'	Residual inner friction angle for base material	Degrees	$0 \leq \phi' \leq \phi$
C5	c'	Residual cohesion for base material	Force/Length ²	$0 \leq c' \leq c$

To define the optional tension Rankine yield surface for the base material, define the material data table (**TB**,JROCK,,,,RCUT) and input the following constants (**TB**DATA):

Table 4.14: Tension Yield Surface for the Base Material (Optional)

Constant	Meaning	Property	Unit	Range
C1	T	Initial tensile strength	Force/Length ²	$T > 0$
C2	T'	Residual tensile strength	Force/Length ²	$0 < T' \leq T$

To define the optional residual strength coupling, define the material data table (**TB,JROCK,,,,RSC**) and input the following constant (**TB,DATA**):

Table 4.15: Residual Strength Coupling for the Base Material (Optional)

Constant	Meaning	Property	Unit	Range
C1	R_{SC}	Residual strength coupling flag	--	$R_{SC}=0,1,2,3$

4.9.4.3.2. Part 2: Defining the Joints

You can define up to four joints. Each joint requires failure plane yield and flow-potential parameters (**TB,JROCK,,,,FPLANE**). Defining this table creates a new set of joint parameters.

After defining a new joint, you can then specify the optional failure plane tensile strength (**TB,JROCK,,,,FTCUT**) and the failure plane orientation (**TB,JROCK,,,,FORIE**).

To initiate a new joint set and specify the failure plane yield and flow-potential parameters:

1. Define the material data table (**TB,JROCK,,,,FPLANE**).
2. Input the required joint constants (**TB,DATA**).

Table 4.16: Jointed Rock Constants for the Joint

Constant	Meaning	Property	Unit	Range
C1	ϕ_j	Inner friction angle for joint	Degrees	$0 \leq \phi_j < 90$
C2	c_j	Initial cohesion for joint	Force/Length ²	$c_j \geq 0$
C3	ψ_j	Dilatancy angle for joint	Degrees	$0 \leq \psi_j \leq \phi_j$
C4	ϕ'_j	Residual inner friction angle for joint	Degrees	$0 \leq \phi'_j \leq \phi_j$
C5	c'_j	Residual cohesion for joint	Force/Length ²	$0 \leq c'_j \leq c_j$

To define the optional tension yield surface for the joint, define the material data table (**TB,JROCK,,,,FTCUT**) and input the following constants (**TB,DATA**):

Table 4.17: Tension Yield Surface for the Joint (Optional)

Constant	Meaning	Property	Unit	Range
C1	T_j	Initial tensile strength for joint	Force/Length ²	$T_j > 0$
C2	T'_j	Residual tensile strength for joint	Force/Length ²	$0 < T'_j \leq T_j$

To define the optional orientation of the failure plane for the joint, define the material data table (**TB,JROCK,,,,FORIE**) and input the following constants (**TB,DATA**):

Table 4.18: Orientation of the Joint Failure Plane (Optional)

Constant	Meaning	Property	Unit
C1	α_j	Reorientation of joint about the negative z axis	Degrees

Constant	Meaning	Property	Unit
C2	β_j	Reorientation of joint about the new y axis	Degrees

Define temperature- or field-dependent data for the data tables via the **TBTEMP** or **TBFIELD** commands, respectively.

Example 4.30: Defining the Jointed Rock Model with an Elastic-Plastic (Mohr-Coulomb) Base

```

/prep7
! ELASTIC CONSTANTS
MP,EX,1,20.0E9
MP,NUXY,1,0.2

! Base Mohr Coulomb yield surface

phi=30      ! friction angle
c=8.7e6     ! cohesion
psi=15      ! dilatancy angle
phir=20     ! residual friction angle
cr=0.8*c    ! residual cohesion

sigt=3e6    ! tensile strength
sigtr=0.8*sigt ! residual tensile strength

TB,JROCK,1,,,BASE
TB,DATA,1,phi,c,psi,phir,cr

TB,JROCK,1,,,RCUT
TB,DATA,1,sigt,sigtr

TB,JROCK,1,,,RSC
TB,DATA,1,0

! Joint 1

phi_j=25    ! friction angle
c_j=1E6     ! cohesion
psi_j=25.0  ! dilatancy angle
phir_j=25   ! residual friction angle
cr_j=0.8*c_j ! residual cohesion

alpha_j=-90 ! negative rotation about element Z-axis
beta_j=45   ! positive rotation about plane Y-axis

TB,JROCK,1,,,FPLANE
TB,DATA,1,phi_j,c_j,psi_j,phir_j,cr_j

TB,JROCK,1,,,FORIE
TB,DATA,1,alpha_j,beta_j

! Joint 2

phi_j=25    ! friction angle
c_j=1E6     ! cohesion
psi_j=25.0  ! dilatancy angle
phir_j=25   ! residual friction angle
cr_j=0.8*c_j ! residual cohesion

sigt_j=0.5E6 ! tensile strength
sigtr_j=0.1E6 ! residual tension strength

alpha_j=-45 ! negative rotation about element Z-axis
beta_j=45   ! positive rotation about plane Y-axis

TB,JROCK,1,,,FPLANE
TB,DATA,1,phi_j,c_j,psi_j,phir_j,cr_j

```

```
TB, JROCK, 1, , , FTCUT
TBDATA, 1, sigt_j, sigtr_j

TB, JROCK, 1, , , FORIE
TBDATA, 1, alpha_j, beta_j
```

4.9.5. Drucker-Prager Concrete

Single-surface Drucker-Prager models often do not represent the large differences in tensile and compressive behavior of concrete. To represent the weak tension behavior, you can use either of the following:

- A second Drucker-Prager yield surface
- A composite surface consisting of a Rankine tension failure surface and a Drucker-Prager surface in compression

The Drucker-Prager yield surfaces are similar to the linear form of the [Extended Drucker-Prager \(p. 55\)](#) yield surface.

Joints represent failure along planes of weakness, such as geometric stress concentrations or inhomogeneous material regions. The Drucker-Prager concrete model can be combined with the anisotropic [Mohr-Coulomb \(p. 136\)](#) model to represent joints in the concrete material. (The anisotropic Mohr-Coulomb model is the same as the [jointed rock \(p. 145\)](#) failure plane model.) The composite Drucker-Prager and Rankine surface cannot be used with joints.

The following topics about the Drucker-Prager concrete material model are available:

[4.9.5.1. Yield Surfaces and Flow Potentials](#)

[4.9.5.2. Defining the Drucker-Prager Concrete Model](#)

Also see [Load-Limit Analysis of a Reinforced Concrete Slab in the Technology Demonstration Guide](#).

4.9.5.1. Yield Surfaces and Flow Potentials

The following topics are available:

[4.9.5.1.1. Tension and Tension-Compression Surfaces](#)

[4.9.5.1.2. Compression Surface](#)

4.9.5.1.1. Tension and Tension-Compression Surfaces

For loading in tension and tension-compression, you can use either a Drucker-Prager yield surface or a Rankine tension failure surface to define the yield condition.

Drucker-Prager Yield Surface

The tension and tension-compression Drucker-Prager yield surface is given by:

$$f_{DP_t} = \frac{\sigma_e}{\sqrt{3}} + \beta_t \sigma_m - \sigma_{Yt}$$

where β_t and σ_{Yt} are constants defined by the uniaxial tensile strength R_t and uniaxial compressive strength R_c :

$$\beta_t = \frac{\sqrt{3}(R_c\Omega_c - R_t\Omega_t)}{R_c\Omega_c + R_t\Omega_t}$$

$$\sigma_{yt} = \frac{2R_c\Omega_c R_t\Omega_t}{\sqrt{3}(R_c\Omega_c + R_t\Omega_t)}$$

Ω_c and Ω_t are hardening/softening functions in compression and tension, which depend on the stress σ and hardening variables κ . For a definition of κ and the specific forms of the hardening/softening functions, see [Hardening, Softening and Dilatation \(HSD\) Behavior](#) (p. 157).

where δ_c is the compression dilatancy parameter.

For the Drucker-Prager yield surface in tension and tension-compression, the flow potential is:

$$Q_{DP_t} = \frac{\sigma_e}{\sqrt{3}} + \delta_t \beta_t \sigma_m$$

Rankine Tension Failure Surface

The Rankine tension failure surface is useful for representing the brittle tensile behavior of concrete. The surface defines yielding when the largest principal stress exceeds the tensile strength and is given by:

$$f_R = \sigma_m + \frac{2}{3} \sigma_e \sin\left(\theta + \frac{2}{3}\pi\right) - T\Omega_t$$

where T is the tensile strength and

$$\sigma_m = \frac{\sigma_{11} + \sigma_{22} + \sigma_{33}}{3}$$

$$\sigma_e = \sqrt{3J_2}$$

$$\sin(3\theta) = -\frac{3\sqrt{3}}{2} \frac{J_3}{\sqrt{J_2^3}}$$

The stress invariants are:

$$J_2 = \frac{1}{6} \left((\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 \right) + \sigma_{12}^2 + \sigma_{23}^2 + \sigma_{13}^2$$

$$J_3 = \det(\sigma - I\sigma_m)$$

The flow potential is the Rankine surface when there is only one positive principal stress. When more than one principal stress is positive, a smooth approximation of the Rankine surface is used as the potential surface, resulting in nonassociated flow.

4.9.5.1.2. Compression Surface

For compressive loading, the Drucker-Prager yield surface is:

$$f_{DPc} = \frac{\sigma_e}{\sqrt{3}} + \beta_c \sigma_m - \sigma_{yc} \Omega_c$$

where the constants β_c and σ_{Yc} are related to the biaxial compressive strength R_b and uniaxial compressive strength R_c by:

$$\beta_c = \frac{\sqrt{3}(R_b - R_c)}{2R_b - R_c}$$

$$\sigma_{Yc} = \frac{R_b R_c}{\sqrt{3}(2R_b - R_c)}$$

The flow potential for yielding in compression is:

$$Q_{DPc} = \frac{\sigma_e}{\sqrt{3}} + \delta_c \beta_c \sigma_m$$

where δ_c the compression dilatancy parameter.

Figure 4.40: Composite Surface with Drucker-Prager Compression and Drucker-Prager Tension Yield Surfaces

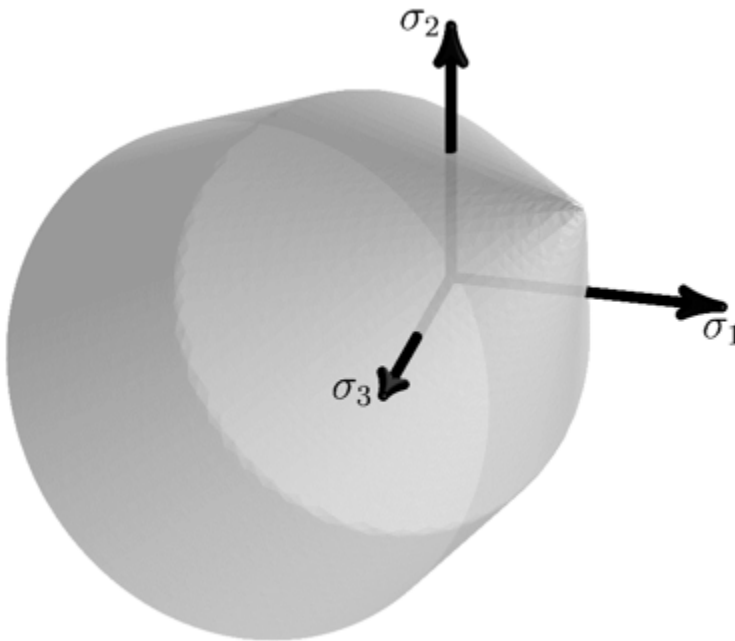
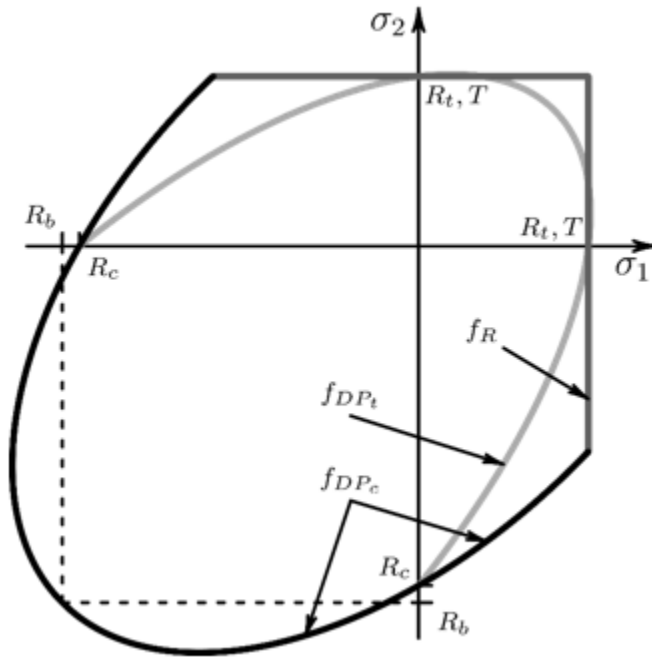


Figure 4.41: 2-D Yield Surfaces Showing Drucker-Prager and Rankine Surfaces

4.9.5.2. Defining the Drucker-Prager Concrete Model

To use the Drucker-Prager concrete model:

1. Define the isotropic or anisotropic elastic behavior (via **MP** commands or via the elastic material data table [**TB**,ELASTIC]).
2. Define the Drucker-Prager concrete yield-strength parameters (**TB**,CONCR,,,,DP).
3. *Optional:* Define the Rankine tension parameter (**TB**,CONCR,,,,RCUT).

The following table shows the Drucker-Prager yield strength parameters (**TB**,CONCR,,,,DP):

Table 4.19: Drucker-Prager Concrete Model Constants

Constant	Meaning	Property	Unit	Range
C1	R_c	Uniaxial compressive strength	Force/Length ²	$R_c > R_t$
C2	R_t	Uniaxial tensile strength Note: This constant is not used if a Rankine tension table is defined	Force/Length ²	$R_t > 0$
C3	R_b	Biaxial compressive strength	Force/Length ²	$R_b > R_c$

To use the Rankine tension surface, define the material data table (**TB,CONCR,,,,RCUT**) and input the uniaxial tension strength:

Table 4.20: Rankine Surface Constant

Constant	Meaning	Property	Unit	Range
C1	T	Uniaxial tensile strength	Force/Length ²	$T > 0$

When using the composite Drucker-Prager with Drucker-Prager tension surface, dilatancy parameters are optional. The composite Drucker-Prager with Rankine surface requires a compression dilatancy parameter.

If you define dilatancy parameters, the following considerations apply:

- You must also define the [hardening, softening, and \(where applicable\) dilatation \(HSD\)](#) (p. 157) behavior.
- You cannot define [failure joints](#) (p. 165), as they are valid only in the perfectly plastic base Drucker-Prager concrete model with associated flow (**TB,CONCR,,,,DP**).

To define dilatancy parameters, define the material data table (**TB,CONCR,,,,DILA**) and input the following constants (**TB,DATA**):

Table 4.21: Drucker-Prager Dilatancy Parameters

Constant	Meaning	Property	Unit	Range
C1	δ_t	Tensile dilatancy parameter Note: This constant is not used if a Rankine tension table is defined	--	$0 < \delta_t \leq 1$
C2	δ_c	Compression dilatancy parameter	--	$0 < \delta_c \leq 1$

The following additional topics are available for defining the Drucker-Prager concrete material model:

[4.9.5.2.1. Hardening, Softening and Dilatation \(HSD\) Behavior](#)

[4.9.5.2.2. Representing Weak Joints in Concrete](#)

[4.9.5.2.3. Example Drucker-Prager Concrete Definitions](#)

4.9.5.2.1. Hardening, Softening and Dilatation (HSD) Behavior

Several HSD behavior models are available:

[4.9.5.2.1.1. Exponential HSD Model \(TB,CONCR,,,,HSD2\)](#)

[4.9.5.2.1.2. Steel Reinforcement HSD Model \(TB,CONCR,,,,HSD4\)](#)

[4.9.5.2.1.3. Fracture Energy HSD Model \(TB,CONCR,,,,HSD5\)](#)

[4.9.5.2.1.4. Linear HSD Model \(TB,CONCR,,,,HSD6\)](#)

When using the Drucker-Prager tension surface, you can use any of the HSD models. When using the Rankine tension surface, the linear, exponential and fracture energy HSD models are available. If no HSD model is defined, the program uses perfect plasticity.

The hardening and softening behavior of the yield surfaces is defined by Ω_t and Ω_c , respectively. In the case of the [steel reinforcement \(p. 160\)](#) HSD model, you can also specify the evolution of the tensile dilatation δ_t .

The Ω_t and Ω_c functions evolve based on the hardening variable κ . The general form of the hardening variable evolution is:

$$d\kappa = \sum_n d\lambda^n \left(\frac{1}{R\Omega} \boldsymbol{\sigma} \cdot \frac{\partial Q^n}{\partial \boldsymbol{\sigma}} \right)$$

where:

n = number of active yield surfaces

R = strength parameter

Ω = hardening or softening function value

$d\lambda^n$ = magnitude of the plastic strain increment for each active yield surface.

Except in special cases, the hardening variable is different from the equivalent plastic strain.

To help overcome mesh-dependent softening behavior, the exponential softening model for the tension yield function and the fracture energy model for the compression yield function are normalized with an effective element length L_i . For 3-D elements, the effective element length is the cube root of the integration point volume. For 2-D elements, it is the square root of the integration point volume. For 1-D elements, the effective element length is the integration point volume.

With the HSD models, the total energy dissipated in a localized failure or crack band approaches the area-specific fracture energy [5].

4.9.5.2.1.1. Exponential HSD Model (TB,CONCR,,,,HSD2)

The hardening yield function in compression, Ω_c , for $\kappa < \kappa_{cm}$ is given by the power-hardening-law function ([Equation 4.44 \(p. 163\)](#)).

The softening yield function in the range $\kappa_{cm} < \kappa < \kappa_{cu}$ is given by:

$$\Omega_c = 1 - (1 - \Omega_{cu}) \left(\frac{\kappa - \kappa_{cm}}{\kappa_{cu} - \kappa_{cm}} \right)^2$$

The softening yield function in the range $\kappa > \kappa_{cu}$ is given by:

$$\Omega_c = \Omega_{cr} + (\Omega_{cu} - \Omega_{cr}) \exp \left(2 \frac{\Omega_{cu} - 1}{\kappa_{cu} - \kappa_{cm}} \cdot \frac{\kappa - \kappa_{cu}}{\Omega_{cu} - \Omega_{cr}} \right)$$

The yield function in tension, Ω_t , is given by an exponential softening function where the volumetric energy dissipated in softening is proportional to the Mode I area-specific fracture energy in tension G_{ft} :

$$\Omega_t = \exp\left(-\frac{\kappa}{a_t}\right) \quad (4.42)$$

where:

$$a_t = \frac{g_{ft}}{R_t}$$

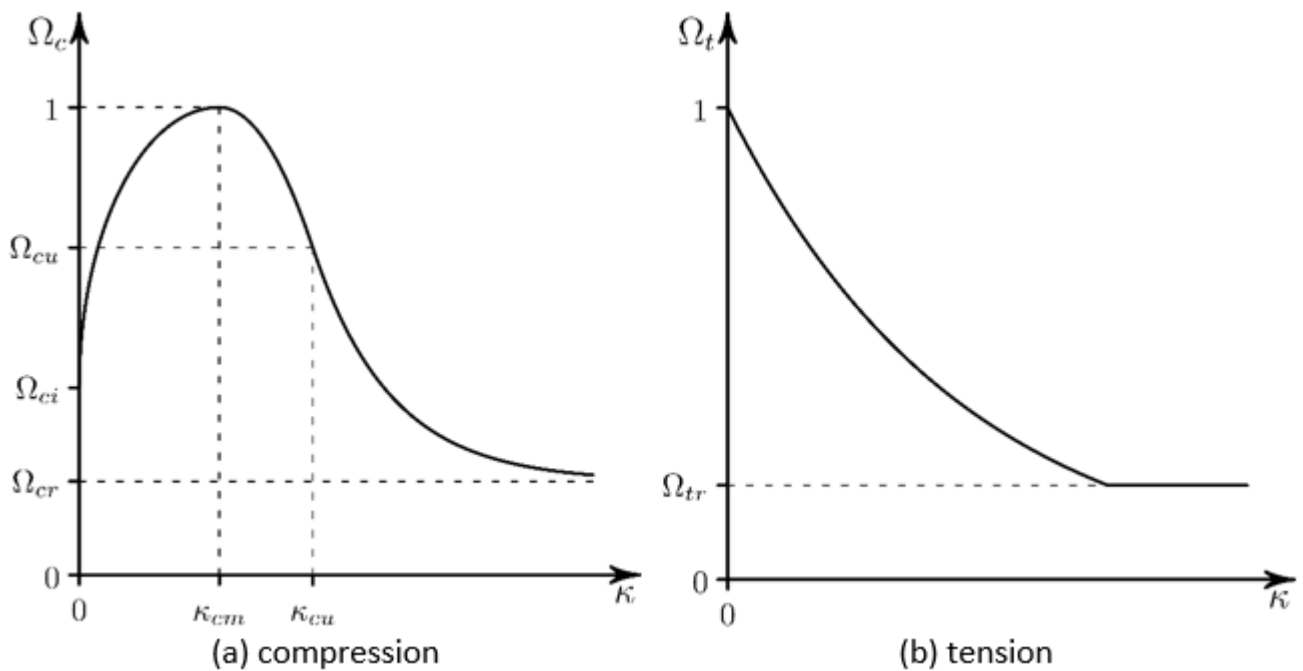
$$g_{ft} = \max\left(\frac{G_{ft}}{L_i}, \frac{R_t^2}{E}\right)$$

where L_i is the effective element length and E is the Young's Modulus, giving the following relation for energy dissipated during softening of the tension yield function:

$$\int_0^{\infty} \Omega_t d\kappa = \frac{g_{ft}}{R_t}$$

To limit the softening, set the relative residual stress level Ω_{tr} to give perfect plasticity when the yield function (Equation 4.42 (p. 159)) is less than Ω_{tr} .

Figure 4.42: Exponential Softening in Compression and Tension



To define exponential softening, define the material data table (TB,CONCR,,,HSD2) and input the following constants (TBDATA):

Table 4.22: Exponential HSD Constants

Constant	Meaning	Property	Unit	Range
C1	κ_{cm}	Plastic strain at uniaxial compressive strength	--	$0 < \kappa_{cm} < 0.02$

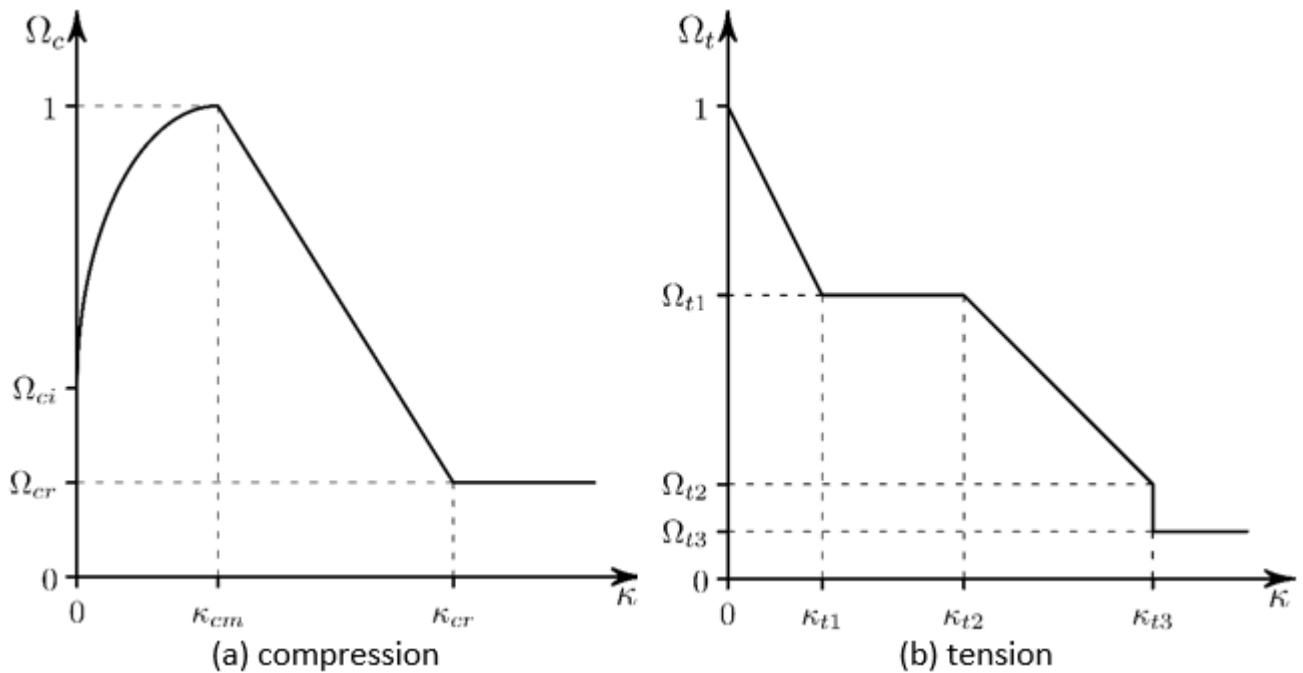
Constant	Meaning	Property	Unit	Range
C2	κ_{cu}	Plastic strain at transition from power law to exponential softening	--	$\kappa_{cr} > \kappa_{cm}$
C3	Ω_{ci}	Relative stress at start of nonlinear hardening	--	$0 \leq \Omega_{ci} \leq 1$
C4	Ω_{cu}	Residual relative stress at κ_{cu}	--	$\Omega_{cr} < \Omega_{cu} \leq 1$
C5	Ω_{cr}	Residual compressive relative stress	--	$0 < \Omega_{cr} < \Omega_{cu}$
C6	G_{ft}	Mode I area-specific fracture energy	Force/Length	$G_{ft} > 0$
C7	Ω_{tr}	Residual tensile relative stress	--	$0 < \Omega_{tr} \leq 1$

4.9.5.2.1.2. Steel Reinforcement HSD Model (TB,CONCR,,,,HSD4)

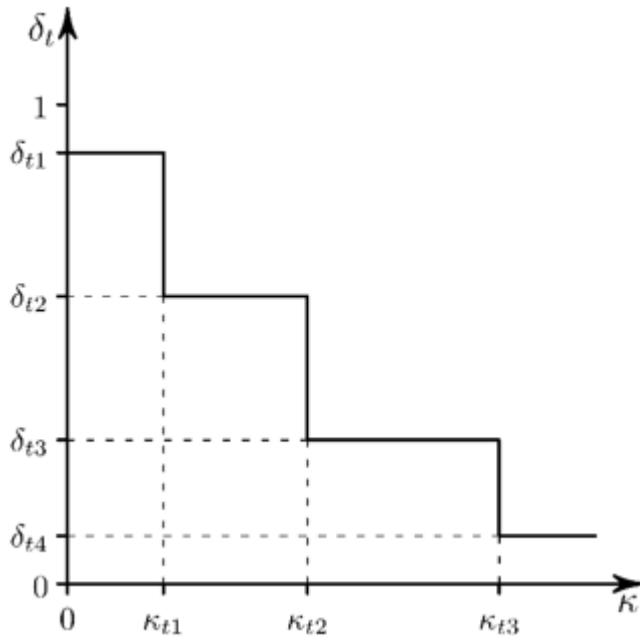
The yield function in compression, Ω_c , is given by the nonlinear hardening function and a [linear softening function](#) (p. 163), with the ultimate effective plastic strain defined as $\kappa_{cr}=0.02$.

The yield function in tension, Ω_t , is defined by a multilinear law for the residual relative strength of steel reinforcement. The multilinear softening is defined by the relative stress vs. effective plastic strain points defined in this figure:

Figure 4.43: Steel Reinforcement Softening in Compression and Tension



The dilatancy factor in tension, δ_t , is defined by a multilinear function. The points defining the dilatancy factor as a function of effective plastic strain are shown in this figure:

Figure 4.44: Steel Reinforcement Dilatancy in Tension

The parameter for tension dilatancy (entered in the dilatancy table [TB,CONCR,,,DILA]) is not used. The compression dilatancy is constant.

To define the steel reinforcement HSD model, define the material data table (TB,CONCR,,,HSD4) and input the following constants (TBDATA):

Table 4.23: Steel Reinforcement HSD Constants

Constant	Meaning	Property	Unit	Range
C1	κ_{cm}	Plastic strain at uniaxial compressive strength	--	$0 < \kappa_{cm} < 0.02$
C2	Ω_{ci}	Relative stress at onset of nonlinear hardening	--	$0 \leq \Omega_{ci} \leq 1$
C3	Ω_{cr}	Residual compressive relative stress	--	$0 < \Omega_{cr} \leq 1$
C4	κ_{t1}	Softening plastic strain point 1	--	$0 < \kappa_{t1}$
C5	κ_{t2}	Softening plastic strain point 2	--	$\kappa_{t1} < \kappa_{t2} < \kappa_{t3}$
C6	κ_{t3}	Softening plastic strain point 3	--	$\kappa_{t2} < \kappa_{t3}$
C7	Ω_{t1}	Residual tensile relative stress point 1	--	$0 < \Omega_{t1} \leq 1$
C8	Ω_{t2}	Residual tensile relative stress point 2	--	$0 < \Omega_{t2} \leq 1$
C9	Ω_{t3}	Residual tensile relative stress point 3	--	$0 < \Omega_{t3} \leq 1$
C10	δ_{t1}	Initial tensile dilatancy parameter	--	$0 < \delta_{t1} \leq 1$
C11	δ_{t2}	Tensile dilatancy parameter point 1	--	$0 < \delta_{t2} \leq 1$

Constant	Meaning	Property	Unit	Range
C12	δ_{t3}	Tensile dilatancy parameter point 2	--	$0 < \delta_{t3} \leq 1$
C13	δ_{t4}	Tensile dilatancy parameter point 3	--	$0 < \delta_{t4} \leq 1$

4.9.5.2.1.3. Fracture Energy HSD Model (TB,CONCR,,,,HSD5)

The yield function in compression, Ω_c , is given by a nonlinear hardening function and a volumetric fracture energy based softening function. The hardening from $\Omega_c = \Omega_{ci}$ to $\Omega_c = 1$ is given above in Equation 4.44 (p. 163). Softening begins to occur at $\kappa = \kappa_{cm}$ and the yield function is:

$$\Omega_c = \frac{1}{a_c (\kappa^2 - 2\kappa_{cm}\kappa + \kappa_{cm}^2) + 1} \quad (4.43)$$

where:

$$a_c = \left(\frac{\pi}{2} \frac{R_c}{g_{fc}} \right)^2$$

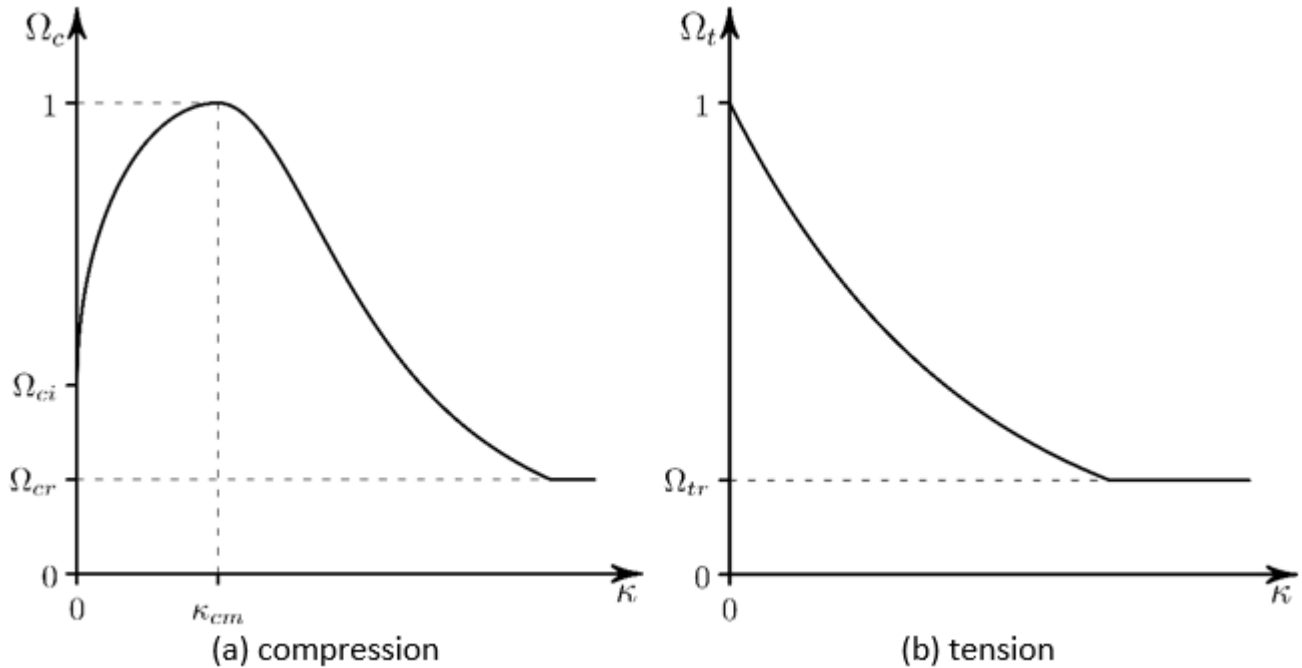
$$g_{fc} = \max \left(\frac{G_{fc}}{L_i}, \frac{R_c^2}{E} \right)$$

where L_i is the effective element length and E is the Young's Modulus.

Then, $\int_{\kappa_{cm}}^{\infty} \Omega_c d\kappa = \frac{g_{fc}}{R_c}$, giving a constant volumetric energy dissipated during softening proportional to the Mode I area-specific fracture energy in compression G_{fc} .

The relative residual stress level Ω_{cr} can be set to limit the softening and give perfect plasticity when the yield function (Equation 4.43 (p. 162)) is less than Ω_{cr} .

The yield function in tension, Ω_t , is given by the exponential softening function described in Exponential HSD Model (TB,CONCR,,,,HSD2) (p. 158).

Figure 4.45: Fracture Energy Softening in Compression and Tension

To define fracture energy softening, define the material data table (**TB,CONCR,,,HSD5**) and input the following constants (**TB,DATA**):

Table 4.24: Fracture Energy HSD Constants

Constant	Meaning	Property	Unit	Range
C1	κ_{cm}	Plastic strain at uniaxial compressive strength	--	$0 < \kappa_{cm}$
C2	Ω_{ci}	Relative stress at onset of nonlinear hardening	--	$0 \leq \Omega_{ci} \leq 1$
C3	Ω_{cr}	Residual compressive relative stress	--	$0 < \Omega_{cr} < 1$
C4	G_{fc}	Mode 1 area-specific fracture energy in compression	Force/Length	$G_{fc} > 0$
C5	G_{ft}	Mode 1 area-specific fracture energy in tension	Force/Length	$G_{ft} > 0$
C6	Ω_{tr}	Residual tensile relative stress	--	$0 < \Omega_{tr} \leq 1$

4.9.5.2.1.4. Linear HSD Model (**TB,CONCR,,,HSD6**)

The yield function in compression, Ω_c , is given by a nonlinear hardening function and a linear softening function. The relative stress level at the onset of nonlinear hardening is Ω_{ci} with a hardening yield function:

$$\Omega_c = \Omega_{ci} + (1 - \Omega_{ci}) \sqrt{2 \frac{\kappa}{\kappa_{cm}} - \frac{\kappa^2}{\kappa_{cm}^2}} \quad (4.44)$$

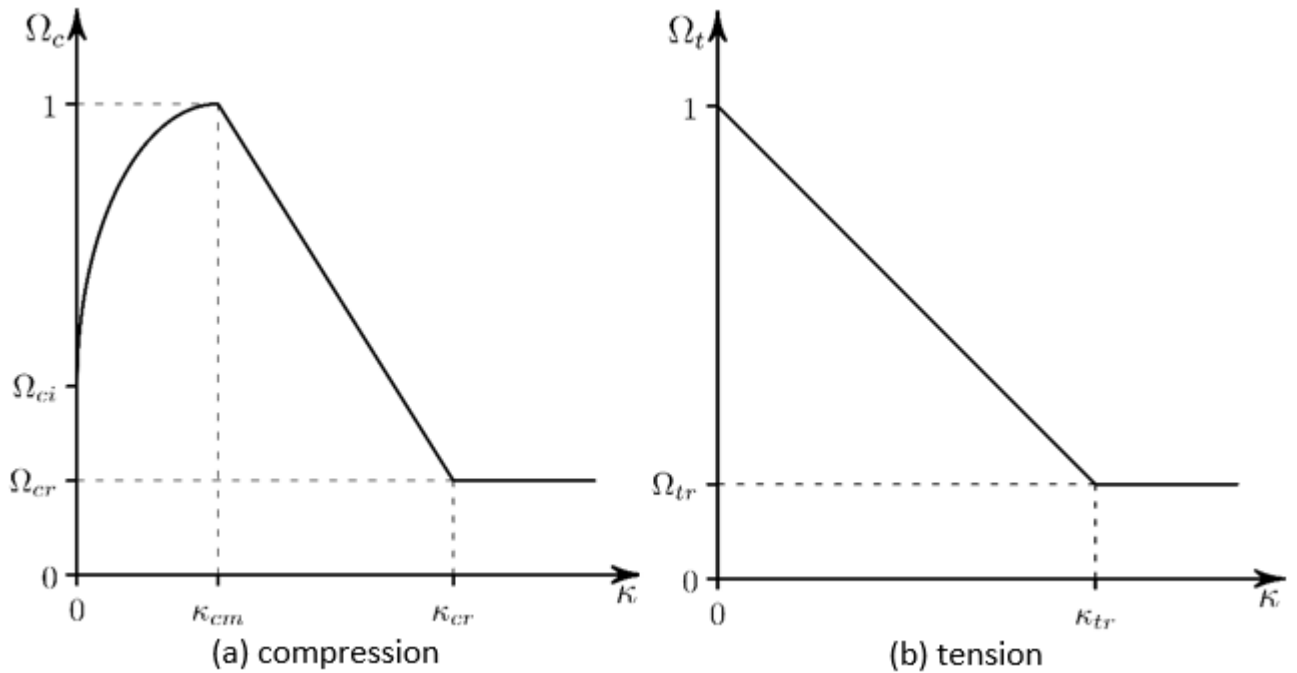
At $\kappa = \kappa_{cm}$, the peak compression strength is reached and softening starts with:

$$\Omega_c = 1 - \frac{1 - \Omega_{cr}}{\kappa_{cr} - \kappa_{cm}} (\kappa - \kappa_{cm})$$

At $\kappa > \kappa_{cr}$, the relative stress level is the residual value.

The yield function in tension, Ω_t , is given by a linear softening function. The relative yield stress is equal to 1 at initial yielding, decreases to the relative residual stress Ω_{tr} when the effective plastic strain is κ_{tr} , and is constant for $\kappa > \kappa_{tr}$.

Figure 4.46: Linear Softening in Compression and Tension



To define linear softening, define the material data table (**TB,CONCR,,,HSD6**) and input the following constants (**TBDATA**):

Table 4.25: Linear HSD Constants

Constant	Meaning	Property	Unit	Range
C1	κ_{cm}	Plastic strain at uniaxial compressive strength	--	$0 < \kappa_{cm} < \kappa_{cr}$
C2	κ_{cr}	Ultimate effective plastic strain in compression	--	$\kappa_{cr} > \kappa_{cm}$
C3	Ω_{ci}	Relative stress at onset of nonlinear hardening	--	$0 \leq \Omega_{ci} \leq 1$
C4	Ω_{cr}	Residual compressive relative stress	--	$0 < \Omega_{cr} \leq 1$
C5	κ_{tr}	Plastic strain limit in tension	--	$\kappa_{tr} \geq 0$
C6	Ω_{tr}	Residual tensile relative stress	--	$0 < \Omega_{tr} \leq 1$

4.9.5.2.2. Representing Weak Joints in Concrete

You can combine the Drucker-Prager concrete model with failure planes to represent weak joints in the concrete. The failure planes are represented by anisotropic Mohr-Coulomb yield surfaces on oriented planes. The process is described in [Defining the Joints \(p. 151\)](#) in the [Jointed Rock \(p. 145\)](#) material model discussion.

Important:

You can define joints in the Drucker-Prager concrete model only with the elastic, perfectly plastic base model with associated flow (**TB,CONCR,,,,DP**). Joints are not applicable when specifying the optional Drucker-Prager dilatancy parameters (**TB,CONCR,,,,DILA**).

You can define up to four joints. Each joint requires failure plane yield- and flow-potential parameters (**TB,CONCR,,,,FPLANE**). Each joint definition creates a new set of joint parameters.

After defining a new joint, you can then specify the optional failure plane tensile strength (**TB,CONCR,,,,FTCUT**) and the failure plane orientation (**TB,CONCR,,,,FORIE**).

To initiate a new joint parameter set and specify the failure plane yield and flow-potential parameters:

1. Define the material data table (**TB,CONCR,,,,FPLANE**).
2. Input the required joint constants (**TB,DATA**).

Table 4.26: Jointed Rock Constants for the Joint

Constant	Meaning	Property	Unit	Range
C1	ϕ_j	Inner friction angle for joint	Degrees	$0 \leq \phi_j < 90$
C2	c_j	Initial cohesion for joint	Force/Length ²	$c_j \geq 0$
C3	ψ_j	Dilatancy angle for joint	Degrees	$0 \leq \psi_j \leq \phi_j$
C4	ϕ'_j	Residual inner friction angle for joint	Degrees	$0 \leq \phi'_j \leq \phi_j$
C5	c'_j	Residual cohesion for joint	Force/Length ²	$0 \leq c'_j \leq c_j$

To define the optional tension yield surface for the joint, define the material data table (**TB,CONCR,,,,FTCUT**) and input the following constants (**TB,DATA**):

Table 4.27: Tension Yield Surface for the Joint (Optional)

Constant	Meaning	Property	Unit	Range
C1	T_j	Initial tensile strength for joint	Force/Length ²	$T_j > 0$
C2	T'_j	Residual tensile strength for joint	Force/Length ²	$0 < T'_j \leq T_j$

To define the optional orientation of the failure plane for the joint, define the material data table (TB,CONCR,,,,FORIE) and input the following constants (TBDATA):

Table 4.28: Orientation of the Joint Failure Plane (Optional)

Constant	Meaning	Property	Unit
C1	α_j	Reorientation of joint about the negative z axis	Degrees
C2	β_j	Reorientation of joint about the new y axis	Degrees

4.9.5.2.3. Example Drucker-Prager Concrete Definitions

Example 4.31: Defining the Drucker-Prager Concrete Model with Dilatancy Parameters

```
/prep7
! ELASTIC CONSTANTS
MP,EX,1,20.0E9
MP,NUXY,1,0.2

! Base Drucker-Prager concrete
Rc=30.0E6      ! uniaxial compressive strength
Rt=3.0E6       ! uniaxial tensile strength
Rb=36.0E6      ! biaxial compressive strength

delta_t=0.25 ! dilatancy factor tension
delta_c=1.0  ! dilatancy factor compression

! Linear hardening/softening
kappa_cm=0.0025-Rc/20.0E9
kappa_cr=0.0025
omega_ci=0.33
omega_cr=0.1

kappa_tr=0.0005
omega_tr=0.2

TB,CONCR,1,,,DP
TBDATA,1,Rc,Rt,Rb

TB,CONCR,1,,,DILA
TBDATA,1,delta_t,delta_c

TB,CONCR,1,,,HSD6
TBDATA,1,kappa_cm,kappa_cr,omega_ci,omega_cr,kappa_tr,omega_tr
```

Example 4.32: Defining the Drucker-Prager Concrete Model with Failure Joints

```
/prep7
! ELASTIC CONSTANTS
MP,EX,1,20.0E9
MP,NUXY,1,0.2

! Base Drucker-Prager concrete
Rc=30.0E6      ! uniaxial compressive strength
Rt=3.0E6       ! uniaxial tensile strength
Rb=36.0E6      ! biaxial compressive strength

TB,CONCR,1,,,DP
TBDATA,1,Rc,Rt,Rb

! Joint 1

phi_j=25      ! friction angle
c_j=1E6       ! cohesion
psi_j=25.0    ! dilatancy angle
```



```

phir_j=25      ! residual friction angle
cr_j=0.8*c_j   ! residual cohesion

alpha_j=-45    ! negative rotation about element Z-axis
beta_j=45      ! positive rotation about plane Y-axis

TB, CONCR, 1, , , FPLANE
TB, DATA, 1, phi_j, c_j, psi_j, phir_j, cr_j

TB, CONCR, 1, , , FORIE
TB, DATA, 1, alpha_j, beta_j

! Joint 2

phi_j=25      ! friction angle
c_j=1E6       ! cohesion
psi_j=25.0    ! dilatancy angle
phir_j=25     ! residual friction angle
cr_j=0.8*c_j  ! residual cohesion

sigt_j=0.5E6  ! tensile strength
sigtr_j=0.1E6 ! residual tension strength

alpha_j=-90   ! negative rotation about element Z-axis
beta_j=45     ! positive rotation about plane Y-axis

TB, CONCR, 1, , , FPLANE
TB, DATA, 1, phi_j, c_j, psi_j, phir_j, cr_j

TB, CONCR, 1, , , FTCUT
TB, DATA, 1, sigt_j, sigtr_j

TB, CONCR, 1, , , FORIE
TB, DATA, 1, alpha_j, beta_j

```

4.9.6. Menetrey-Willam

The Menetrey-Willam constitutive model [1] is based on the Willam-Warnke yield surface [2], incorporating dependence on three independent invariants of the stress tensor.

The Willam-Warnke surface is similar to the [Mohr-Coulomb \(p. 136\)](#) surface, but without the sharp edges that can cause difficulty in the Mohr-Coulomb surface stress solution. It also shares some characteristics with the [Drucker-Prager \(p. 153\)](#) model and can model similar materials. The Menetrey-Willam model, however, is generally better for simulating the behavior of bonded aggregates such as concrete.

The following topics for the Menetrey-Willam model are available:

- [4.9.6.1. Yield Surfaces and Flow Potentials](#)
- [4.9.6.2. Defining the Menetrey-Willam Model](#)
- [4.9.6.3. Hardening and Softening Functions](#)
- [4.9.6.4. Example Menetrey-Willam Definitions](#)
- [4.9.6.5. References](#)

4.9.6.1. Yield Surfaces and Flow Potentials

The material parameters defining the yield function are:

- Yield strengths in uniaxial tension (R_t)

- Uniaxial compression (R_c)
- Biaxial compression (R_b)

Parameter hardening and softening is defined by:

$$\bar{R}_t = R_t \Omega_{tc}$$

$$\bar{R}_c = R_c \Omega_c$$

$$\bar{R}_b = R_b \Omega_c$$

$$\Omega_{tc} = \begin{cases} \Omega_t & \kappa_c \leq \kappa_{cm} \\ \Omega_t \Omega_c & \kappa_c > \kappa_{cm} \end{cases}$$

where κ_{cm} is a material parameter, and Ω_c and Ω_t are the compression and tension-hardening/softening functions that depend on the compression or tension-hardening variables κ_c and κ_t .

The yield surface in Haigh-Westergaard stress coordinates is given by:

$$f_{MW} = \frac{c_2}{c_3} \left[\sqrt{2} \xi + r \rho \right] + \rho^2 - \frac{1}{c_3}$$

where c_2 and c_3 are functions of the material parameters and the hardening softening functions

$$c_2 = \frac{1}{\sqrt{6}} \left[\frac{1}{\bar{R}_c} - \frac{1}{\bar{R}_b} + \frac{\bar{R}_b - \bar{R}_t}{\bar{R}_c^2} \right]$$

$$c_3 = \frac{3}{2} \frac{1}{\bar{R}_c^2}$$

and

$$r = \frac{4(1-e^2)\cos^2\theta + (2e-1)^2}{2(1-e^2)\cos\theta + (2e-1)\sqrt{4(1-e^2)\cos^2\theta + 5e^2 - 4e}}$$

$$e = \frac{1+\epsilon}{2-\epsilon}$$

$$\epsilon = \frac{\bar{R}_t}{\bar{R}_b} \frac{\bar{R}_b^2 - \bar{R}_c^2}{\bar{R}_c^2 - \bar{R}_t^2}$$

The Haigh-Westergaard stress coordinates are:

$$\xi = \frac{1}{\sqrt{3}} I_1$$

$$\rho = \sqrt{2J_2}$$

$$\cos 3\theta = \frac{3\sqrt{3}}{2} \frac{J_3}{\sqrt{J_2^3}}$$

with I_1 the first principal invariant of the stress tensor, and J_2 and J_3 the second and third principal invariants of the deviatoric stress tensor:

$$I_1 = \sigma_{11} + \sigma_{22} + \sigma_{33}$$

$$J_2 = \frac{1}{6} \left((\sigma_{11} - \sigma_{22})^2 + (\sigma_{22} - \sigma_{33})^2 + (\sigma_{33} - \sigma_{11})^2 \right) + \sigma_{12}^2 + \sigma_{23}^2 + \sigma_{13}^2$$

$$J_3 = \det(\sigma - I\sigma_m)$$

The flow potential is:

$$Q_{MW} = \rho^2 + B_g \rho + C_g \xi$$

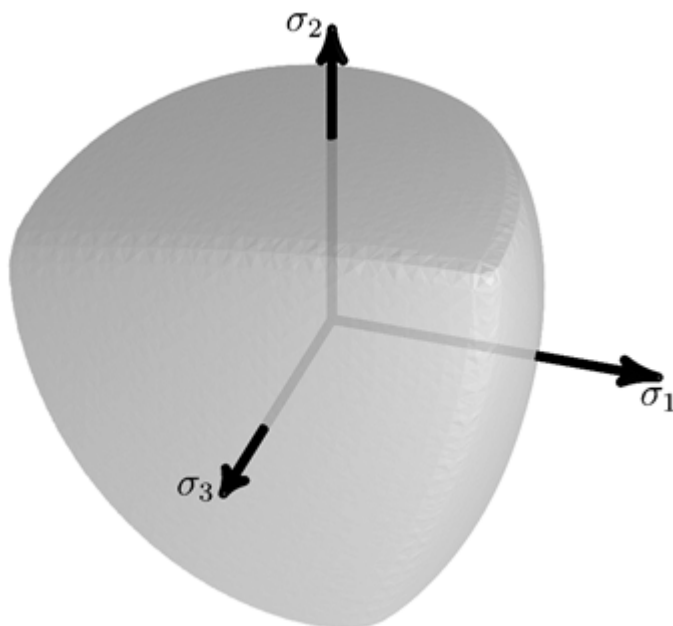
where:

$$B_g = \frac{2\bar{R}_c \tan \psi - \sqrt{2} \bar{R}_t}{\sqrt{3} (1 - \sqrt{2} \tan \psi)}$$

$$C_g = \frac{B_g}{\sqrt{2}} + \frac{2\bar{R}_t}{\sqrt{3}}$$

and ψ is the dilatancy angle. The flow potential gives unassociated plastic flow and results in an unsymmetric consistent material tangent.

Figure 4.47: Yield Surface for the Menetrey-Willam Model



4.9.6.2. Defining the Menetrey-Willam Model

Define the isotropic or anisotropic elastic behavior via **MP** commands or via the elastic table (**TB,ELASTIC**).

To define the Menetrey-Willam yield strength parameters, define the material data table (**TB,CONCRETE,,,MW**) and input the following constants (**TB,DATA**):

Constant	Meaning	Property	Units	Range
C1	R_c	Uniaxial compressive strength	Force/Length ²	$R_c > R_t$
C2	R_t	Uniaxial tensile strength	Force/Length ²	$R_t > 0$
C3	R_b	Biaxial compressive strength	Force/Length ²	$R_b > R_c$

To define the dilatancy angle, define the material data table (**TB,CONCRETE,,,DILA**) and input the following constant (**TB,DATA**):

Constant	Meaning	Property	Units	Range
C1	ψ	Dilatancy angle	Degrees	$\frac{R_t}{\sqrt{2}R_c} < \tan\psi \leq \frac{1}{\sqrt{2}}$

4.9.6.3. Hardening and Softening Functions

The hardening-softening behavior of the yield surfaces is defined by the hardening-softening functions Ω_t and Ω_c . These functions depend on the compression- and tension-hardening variables and which evolve according to the work hardening expressions: [4,5]

$$\dot{\kappa}_c = \frac{\alpha_c}{R_c} \sigma \cdot \dot{\epsilon}^{pl}$$

$$\dot{\kappa}_t = \frac{\alpha_t}{R_t} \sigma \cdot \dot{\epsilon}^{pl}$$

where α_c and α_t are compression and tension weight functions given by:

$$\alpha_c = 1 - \alpha_t$$

$$\alpha_t = \begin{cases} 0 & \tan\alpha < -2 \\ \frac{1}{1 + \exp(-10 \tan\alpha)} & -2 \leq \tan\alpha \leq 2 \\ 1 & \tan\alpha > 2 \end{cases}$$

$$\tan\alpha = \sqrt{6} \frac{\xi}{\rho}$$

Specifying the hardening-softening behavior is optional. If a hardening-softening model is not specified, the material is considered to be elastic-perfectly plastic.

To help overcome mesh-dependent softening behavior, the fracture-energy model is normalized with an effective element length L_i :

- 3-D elements – L_i = cube root of the integration-point volume
- 2-D elements – L_i = square root of the integration-point volume
- 1-D elements – L_i = integration-point volume

With the fracture-energy model, the total energy dissipated in a localized failure or crack band approaches the area-specific fracture energy [3].

4.9.6.3.1. Linear Softening

The yield function in compression, Ω_c , is given by a nonlinear hardening function and a linear softening function. The relative stress level at the start of nonlinear hardening is Ω_{ci} , with a hardening yield function of:

$$\Omega_c = \Omega_{ci} + (1 - \Omega_{ci}) \sqrt{2 \frac{\kappa}{\kappa_{cm}} - \frac{\kappa^2}{\kappa_{cm}^2}} \quad (4.45)$$

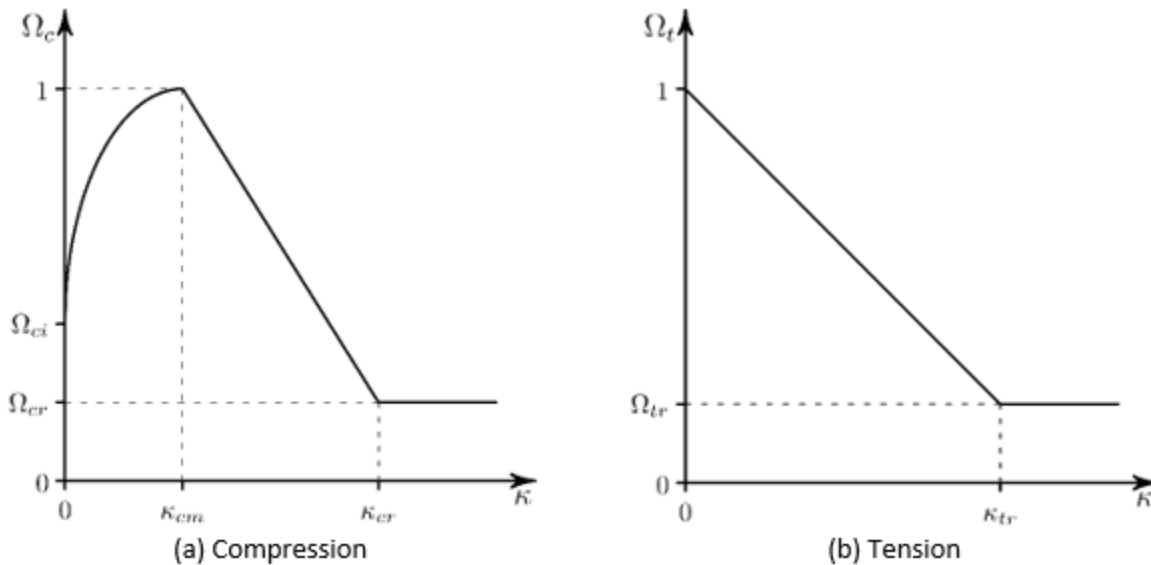
At $\kappa = \kappa_{cm}$, the peak compression strength is reached and softening starts with:

$$\Omega_c = 1 - \frac{1 - \Omega_{cr}}{\kappa_{cr} - \kappa_{cm}} (\kappa - \kappa_{cm}) \quad (4.46)$$

At $\kappa > \kappa_{cr}$, the relative stress level is the residual value Ω_{cr} .

The yield function in tension, Ω_t , is given by a linear softening function. The relative yield stress is equal to 1 at initial yielding, decreases to the relative residual stress Ω_{tr} when the effective plastic strain is κ_{tr} , and is constant for $\kappa > \kappa_{tr}$.

Figure 4.48: Linear Softening in Compression and Tension



To specify linear softening, define the material data table (**TB,CONCRETE,,,HSD6**) and input the following constants (**TB,DATA**):

Constant	Meaning	Property	Units	Range
C1	κ_{cm}	Plastic strain at uniaxial compressive strength	-	$0 < \kappa_{cm} < \kappa_{cr}$
C2	κ_{cr}	Ultimate effective plastic strain in compression	-	$\kappa_{cr} > \kappa_{cm}$
C3	Ω_{ci}	Relative stress at start of nonlinear hardening	-	$0 \leq \Omega_{ci} \leq 1$
C4	Ω_{cr}	Residual compressive relative stress	-	$0 < \Omega_{cr} \leq 1$
C5	κ_{tr}	Plastic strain limit in tension	-	$\kappa_{tr} \geq 0$
C6	Ω_{tr}	Residual tensile relative stress	-	$0 < \Omega_{tr} \leq 1$

4.9.6.3.2. Exponential Softening

The hardening yield function in compression, Ω_c , for $\kappa < \kappa_{cm}$ is given by the power hardening law function in Equation 4.45 (p. 171). The softening yield function in the range $\kappa_{cm} < \kappa < \kappa_{cu}$ is given by:

$$\Omega_c = 1 - (1 - \Omega_{cu}) \left(\frac{\kappa - \kappa_{cm}}{\kappa_{cu} - \kappa_{cm}} \right)^2$$

and for $\kappa > \kappa_{cu}$:

$$\Omega_c = \Omega_{cr} + (\Omega_{cu} - \Omega_{cr}) \exp \left(2 \frac{\Omega_{cu} - 1}{\kappa_{cu} - \kappa_{cm}} \cdot \frac{\kappa - \kappa_{cu}}{\Omega_{cu} - \Omega_{cr}} \right)$$

The yield function in tension, Ω_t , is given by an exponential softening function where the volumetric energy dissipated in softening is proportional to the mode I area specific fracture energy in tension G_{ft} :

$$\Omega_t = \exp \left(-\frac{\kappa}{a_t} \right) \quad (4.47)$$

where:

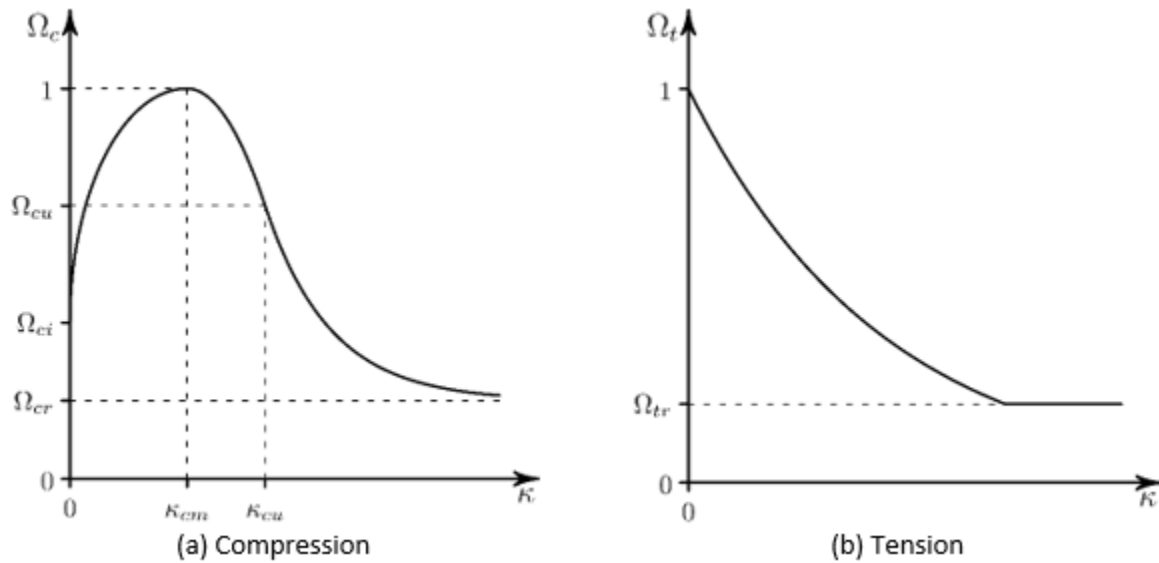
$$a_t = \frac{g_{ft}}{R_t}$$

$$g_{ft} = \max \left(\frac{G_{ft}}{L_i}, \frac{R_t^2}{E} \right)$$

where L_i is the effective element length and E is the Young's Modulus, giving the following relation for energy dissipated during softening of the tension yield function:

$$\int_0^\infty \Omega_t d\kappa = \frac{g_{ft}}{R_t}$$

To limit the softening, set the relative residual stress level Ω_{tr} to give perfect plasticity when the yield function calculated via Equation 4.47 (p. 172) is less than Ω_{tr} .

Figure 4.49: Exponential Softening in Compression and Tension

To define exponential softening, define the material data table (**TB,CONCRETE,,,HSD2**) and input the following constants (**TBDATA**):

Constant	Meaning	Property	Units	Range
C1	κ_{cm}	Plastic strain at uniaxial compressive strength	-	$0 < \kappa_{cm} < \kappa_{cu}$
C2	κ_{cu}	Plastic strain at transition from power law to exponential softening	-	$\kappa_{cu} > \kappa_{cm}$
C3	Ω_{ci}	Relative stress at start of nonlinear hardening	-	$0 \leq \Omega_{ci} \leq 1$
C4	Ω_{cu}	Residual relative stress at κ_{cu}	-	$\Omega_{cr} < \Omega_{cu} \leq 1$
C5	Ω_{cr}	Residual compressive relative stress	-	$0 < \Omega_{cr} < \Omega_{cu}$
C6	G_{ft}	Mode I area-specific fracture energy	Force-length/length ²	$G_{ft} > 0$
C7	Ω_{tr}	Residual tensile relative stress	-	$0 < \Omega_{tr} \leq 1$

4.9.6.4. Example Menetrey-Willam Definitions

Example 4.33: Defining the Menetrey-Willam Concrete Model

```

/prep7
! ELASTIC CONSTANTS
MP,EX,1,20.0E9
MP,NUXY,1,0.2

! base Menetrey-Willam concrete
Rc=30.0E6      ! uniaxial compressive strength
Rt=3.0E6       ! uniaxial tensile strength
Rb=36.0E6      ! biaxial compressive strength

```

```

psi=10          ! dilatancy angle

! linear hardening softening
kappa_cm=0.0025-Rc/20.0E9
kappa_cr=0.0025
omega_ci=0.33
omega_cr=0.1

kappa_tr=0.0005
omega_tr=0.2

TB, CONCRETE, 1, , , MW
TBDATA, 1, Rc, Rt, Rb

TB, CONCRETE, 1, , , DILA
TBDATA, 1, psi

TB, CONCRETE, 1, , , HSD6
TBDATA, 1, kappa_cm, kappa_cr, omega_ci, omega_cr, kappa_tr, omega_tr

```

4.9.6.5. References

1. Menetrey, P. "Numerical Analysis of Punching Failure in Reinforced Concrete Structures." Diss. Ecole Polytechnique Federale de Lausanne, Lausanne, 1994. [Infoscience](#). Web.
2. Willam, K. J. and E. P. Warnke. "Constitutive Models for the Triaxial Behavior of Concrete." *Seminar on Concrete Structures Subjected to Triaxial Stresses. International Association for Bridge and Structural Engineering*. 19 (1975): 1-30.
3. Bazant, Z. P., B. H. Oh. "Microplane Model for Progressive Fracture of Concrete and Rock." *Journal for Engineering Mechanics*. 111 (1985): 559-582.
4. Jirasek, Milan. "Numerical Modeling of Deformation and Failure of Materials." Lecture notes. 2000.
5. Chen, W. F. "Constitutive Equations for Engineering Materials." *Volume 2: Plasticity and Modeling*. Elsevier. 1994.

4.10. Porous Media

Porous media are materials containing pores. The materials are composed of solid-phase and open or closed pores. The solid portion is often called the *skeleton*. This documentation deals primarily with open pores filled with fluid (liquid or gas).

Many natural materials (such as rocks, soil and zeolites), biological materials (such as bones, tissue, wood and cork), and man-made materials (such as cements and ceramics) can be considered porous media. Pore structure geometry is inherently complicated. For engineering purposes, many of their important properties/characters can only be rationalized at a macroscopic scale by considering them to be porous media. Mechanics involving porous media must include mechanical response and fluid flow, requiring a coupled [structural-pore-fluid-diffusion-thermal analysis](#).

The following porous-media topics are available:

- [4.10.1. Fluid Flow and Permeability](#)
- [4.10.2. Porous Media Mechanics](#)
- [4.10.3. Porous Media Material Properties](#)
- [4.10.4. Heat-Transfer Properties](#)

4.10.5. Thermal-Expansion Material Properties

4.10.6. Transient vs. Static Analysis

4.10.7. Partially Saturated Porous Media Flow and Coupled-Pore-Pressure-Thermal (CPT) Damping

4.10.8. Additional Resources

Also see:

- [Porous Media Flow in the *Theory Reference*](#)
- [Initial Degree of Saturation and Relative Permeability Application in the *Advanced Analysis Guide*](#)
- [Surface Subsidence Caused by Reservoir Depletion in the *Technology Demonstration Guide*](#)
- [Normal-Pressure Hydrocephalus Analysis Using Hyperelastic Material in the *Technology Demonstration Guide*](#)

4.10.1. Fluid Flow and Permeability

The flow of water or other liquid through porous media (such as soils) occurs when a difference in the water level exists, causing fluid pressure. Darcy's law defines porous media flow:

$$\mathbf{q}_f = \frac{k_r \mathbf{k}}{\gamma_f} \left(-\nabla p + s_f \gamma_f \mathbf{g}_v \right) \quad (4.48)$$

where:

$\mathbf{q}_f$ = flow flux (Units: Length / Time)

k_r = relative permeability

$\mathbf{k}$ = permeability

s_f = degree of fluid saturation

γ_f = specific weight of fluid

p = pore pressure

$\nabla = \left[\frac{\partial}{\partial x} \quad \frac{\partial}{\partial y} \quad \frac{\partial}{\partial z} \right]^T$ = gradient operator (3-D form shown)

$\mathbf{g}_v$ = gravity load direction (not to be confused with gravity magnitude)

4.10.1.1. Permeability

Permeability is a measure of the porous media's ability to allow fluids (gas or liquid) to flow through it. The higher the permeability of the porous media, the faster fluids can move through it.

Generally, permeability is a 3 x 3 matrix and can be isotropic, orthotropic or anisotropic. To define the porous media material data-table permeability matrix, issue the **TB,PM** command.

For more information, see [Porous Media Material Properties](#) (p. 180).

4.10.2. Porous Media Mechanics

Porous media are multiphase materials containing pores. The void ratio e is defined by:

$$e = V_v / V_s$$

where:

V_v = void volume

V_s = solid volume

Porosity n is defined as:

$$n = V_v / V$$

where V = total volume.

Porosity and void ratio are related by:

$$n = e / (1 + e)$$

The balance equations assume small-strain theory. They are as follows:

- [Momentum-balance equation \(p. 176\)](#)
- [Mass-balance \(continuity\) equation \(p. 177\)](#) for the pore-fluid phase
- [Heat-transfer equation \(p. 179\)](#)

4.10.2.1. Momentum-Balance Equation

The momentum-balance equation for a two-phase medium:

$$\nabla \sigma - \rho_b \ddot{\mathbf{u}} + \gamma_b \mathbf{g} = 0 \quad (4.49)$$

where:

σ = total stress tensor

$$\nabla = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial z} \\ 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial x} & \frac{\partial}{\partial z} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} = \text{matrix differentiation operator (3-D form shown)}$$

$\mathbf{u}$ = displacement

γ_b = bulk specific weight of porous media (defined as $\gamma_b = (1-n)\gamma_s + n\gamma_f$, where γ_s is the solid skeleton specific weight and n is the porosity)

ρ_b = bulk density of porous media

The bulk density of the porous media is entered via the **MP,DENS** or **TB,DENS** commands. When solid and fluid properties are defined (**TB,PM,,,SP** or **FP**) and gravity magnitude is defined (**TB,PM,,,GRAV**), bulk density is calculated as:

$$\rho_b = \frac{\{(1-n)\gamma_s + n s_f \gamma_f\}}{g} = \frac{\gamma_b}{g}$$

where g is the magnitude of gravitational acceleration. A typical value is 9.8m/s^{-2} .

The specific weights of solid and pore fluid are defined via the porous media data table (**TB**,PM). The gravity load direction is defined via the specific weight load setting (**SSOPT**,SFSW).

4.10.2.2. Mass-Balance (Continuity) Equation for the Pore-Fluid Phase

The mass-balance (continuity) equation for the pore-fluid phase is:

$$\nabla \cdot \mathbf{q}_f + \alpha \dot{\varepsilon}_v + \frac{\dot{p}}{Q^*} - s_f \dot{\varepsilon}^{free} = 0 \quad (4.50)$$

where:

ε_v = volumetric strain

Q^* = compressibility parameter

ε^{free} = free strain

T = temperature

The following topics for the mass-balance (continuity) equation are available:

4.10.2.2.1. Compressibility Parameter

4.10.2.2.2. Biot Coefficient

4.10.2.2.3. Effective Stress

4.10.2.2.4. Free-Strain Rate

4.10.2.2.1. Compressibility Parameter

The compressibility parameter Q^* is calculated from the Biot modulus (**TB**,PM,,,,BIOT) and partially saturated parameters:

$$\frac{1}{Q^*} = C_s + s_f \left(\frac{1}{K_m} \right)$$

$$C_s = n \frac{ds_f}{dp}$$

For fully saturated flow the parameters take the following values:

$$C_s = 0, \quad s_f = 1$$

When solid and fluid properties are defined (**TB**,PM,,,,SP or FP), the compressibility parameter is calculated using the solid skeleton and fluid modulus as:

$$\frac{1}{Q^*} = C_s + s_f \left(\frac{n}{K_f} + \frac{(\alpha - n)}{K_s} \right) \quad (4.51)$$

where:

K_s = bulk modulus of skeleton

K_f = bulk modulus of fluid

In this case, the Biot modulus input via **TB,PM,,,,BIOT** is ignored.

4.10.2.2.2. Biot Coefficient

The Biot coefficient is input via the **TB,PM,,,,BIOT** command. However, if solid and fluid properties are defined (**TB,PM,,,,SP/FP**), the Biot coefficient is calculated as:

$$\alpha = 1 - \frac{K_T}{K_S}$$

where:

$$K_T = \frac{\mathbf{I}^T(\mathbf{D})\mathbf{I}}{9} = \text{tangential bulk modulus}$$

$\mathbf{D}$ = material tangent stiffness (from solid skeleton material behavior)

4.10.2.2.3. Effective Stress

The constitutive behavior of the solid skeleton is based on effective stress principal that describes mechanical response of material. The solid part of the model is represented by the effective stress as:

$$\boldsymbol{\sigma}'' = \boldsymbol{\sigma} + \alpha s_f p \mathbf{I} \quad (4.52)$$

where $\boldsymbol{\sigma}'$ is the effective stress tensor. In general, the effective stress is a function of time, strain and other history-dependent solution variables, if any.

The effective stress causes all relevant deformation of the solid skeleton and is related to the elastic strain tensor via Hooke's law:

$$\boldsymbol{\sigma}'' = \mathbf{D} : \boldsymbol{\varepsilon}_{el} = \mathbf{D} : (\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_{pl} - \boldsymbol{\varepsilon}_{th} - \boldsymbol{\varepsilon}_o - \boldsymbol{\varepsilon}_{in}) \quad (4.53)$$

where:

$\mathbf{D}$ = material elastic tangent stiffness

$\boldsymbol{\varepsilon}_{el}$ = elastic strain tensor

$\boldsymbol{\varepsilon}_{pl}$ = plastic strain tensor

$\boldsymbol{\varepsilon}_{th}$ = thermal strain tensor

$\boldsymbol{\varepsilon}_o$ = initial strain tensor

$\boldsymbol{\varepsilon}_{in}$ = inelastic strains other than plasticity

4.10.2.2.4. Free-Strain Rate

The form of the free-strain-rate term $\dot{\boldsymbol{\varepsilon}}^{free}$ depends on the thermal expansion coefficient (CTE) definitions:

- **Only solid skeleton CTE defined:**

$$\varepsilon^{free} = \beta \dot{T}$$

$$\beta = \beta_s = \alpha_x + \alpha_y + \alpha_z$$

where:

β = instantaneous volumetric thermal expansion coefficient

β_s = solid skeleton instantaneous volumetric thermal expansion coefficient

$\alpha_x, \alpha_y, \alpha_z$ = linear instantaneous thermal expansion coefficients

Defining the instantaneous solid skeleton CTEs

The instantaneous thermal expansion coefficients are equal to the secant thermal expansion coefficients if there is no variation of the coefficients during solution (that is, if there is no temperature (or other field-variable) dependence. In that case, define the instantaneous solid skeleton CTEs via **TB,CTE** (secant CTE input). If the **TB,CTE** properties are made temperature- (or other field-variable-) dependent, the program uses the CTE values directly without accounting for conversion to instantaneous values.

To account for temperature dependence of the instantaneous solid skeleton CTEs, define the solid skeleton CTEs via **MP** commands (**MPTEMP** and **MPDATA,CTEX/Y/Z**). In this case, the program uses the instantaneous CTE directly with respect to the mass balance (continuity) equation for the pore-fluid phase (Equation 4.50 (p. 177)). For the momentum-balance equation of the porous medium (Equation 4.49 (p. 176)), which uses the secant CTE, the program converts the instantaneous values to the secant values (p. 21).

• Fluid CTE defined:

$$\varepsilon^{free} = \beta \dot{T}$$

$$\beta = (\alpha - n)\beta_s + n\beta_f = (\alpha - n)(\alpha_x + \alpha_y + \alpha_z) + n\beta_f$$

where:

β_f = fluid-thermal-expansion coefficient (input via **TB,CTE,,,FLUID**)

4.10.2.3. Heat-Transfer Equation

The heat-transfer equation is:

$$(\rho c)_{eff} \dot{T} - \nabla \cdot (\mathbf{D}_T \nabla T) + \nabla \cdot (n s_f \rho_f c_f \mathbf{q}_f T) = 0 \quad (4.54)$$

where:

$\mathbf{D}_T$ = thermal conductivity (Units: Mass * Length * Temperature⁻¹ * Time⁻³)

ρ_f = fluid density calculated as $\rho_f = \frac{\gamma_f}{g}$.

The form of $(\rho c)_{eff}$ depends on the **TB** table definition:

- **Only **TB**, **THERM**, **SPHT** defined:**

$$(\rho c)_{eff} = \rho_b c_s$$

where:

c_s = solid-skeleton specific heat

- ****TB**, **THERM**, **FLSPHT** defined:**

$$(\rho c)_{eff} = (1-n)\rho_s c_s + n s_f \rho_f c_f$$

where:

c_f = fluid specific heat

ρ_s = solid-skeleton density calculated as $\rho_s = \frac{\gamma_s}{g}$

4.10.3. Porous Media Material Properties

The material properties for porous media analysis include solid skeleton and pore-fluid properties. Define material model constants for a porous medium (**TB**,**PM**).

The following material-property options (**TBOPT**) are available for defining porous media:

- Fluid permeability (p. 180) (**PERM**)
- Biot (p. 181) (**BIOT**)
- Solid (p. 181) (**SP**)
- Fluid (p. 182) (**FP**)
- Degree of saturation (p. 182) (**DSAT**)
- Relative permeability (p. 182) (**RPER**)
- Gravity magnitude (p. 182) (**GRAV**)

Define constants (**Cn**) via **TBDATA** (or **TBPT**, as applicable).

4.10.3.1. Permeability (**TB**,**PM**, **TBOPT** = **PERM**)

The permeability properties can be isotropic, orthotropic or anisotropic. The full 3 x 3 permeability matrix is:

$$\mathbf{k} = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{yx} & k_{yy} & k_{yz} \\ k_{zx} & k_{zy} & k_{zz} \end{bmatrix}$$

Define up to nine constants:

Isotropic permeability matrix -- Define k_{xx} only. (The k_{yy} and k_{zz} values are assumed to be k_{xx} .)

Orthotropic permeability matrix -- Enter three constants: k_{xx} , k_{yy} and k_{zz} . The program assigns values of zero to the off-diagonal permeability constants (k_{ij} , $i \neq j$).

Anisotropic permeability matrix -- If you define the first six constants (k_{xx} , k_{yy} , k_{zz} , k_{xy} , k_{xz} , k_{yz}), the program makes the permeability matrix symmetric (that is, $k_{xy}=k_{yx}$, $k_{xz}=k_{zx}$, $k_{yz}=k_{zy}$).

General nonsymmetric anisotropic permeability matrix -- Define all nine constants.

Constant	Meaning	Units
C1	k_{xx}	Length/Time (p. 183)
C2	k_{yy}	
C3	k_{zz}	
C4	k_{xy}	
C5	k_{xz}	
C6	k_{yz}	
C7	k_{yx}	
C8	k_{zx}	
C9	k_{zy}	

All defined constants are based on the element coordinate system. By default, the element coordinate system is the global coordinate system.

4.10.3.2. Biot Coefficient (**TB,PM,,,, TB0PT = BIOT**)

Define the Biot coefficient and Biot modulus.

If the Biot coefficient remains undefined, the program assigns a default value of 1. If the Biot modulus remains undefined, the porous media is assumed to be incompressible.

Constant	Meaning	Property	Units
C1	α	Biot coefficient	Dimensionless
C2	k_m	Biot modulus	Force/Length ²

If solid property and/or fluid property is defined (**TB,PM,,,,SP/FP**), the Biot coefficient and Biot modulus are overwritten with values calculated as described in [Porous Media Mechanics \(p. 175\)](#).

4.10.3.3. Solid Property (**TB,PM,,,, TB0PT = SP**)

Defining either of the available constants is *optional*:

- If the bulk modulus of the solid skeleton is not defined, the solid skeleton is assumed to be incompressible.

- If the specific weight of the solid skeleton is defined, the program considers it to be a body force.

Constant	Meaning	Property	Units
C1	k_s	Bulk modulus of solid skeleton	Force/Length ²
C2	γ_s	Specific weight of solid skeleton	Force/Length ³

4.10.3.4. Fluid Property (TB,PM,,,, TBOPT = FP)

Defining any of the three available constants is *optional*:

- If the bulk modulus of the fluid is not defined, the fluid is assumed to be incompressible.
- If the specific weight of the fluid is defined, the program considers it to be a body force load.

Constant	Meaning	Property	Units
C1	k_f	Bulk modulus of fluid	Force/Length ²
C2	γ_f	Specific weight of fluid	Force/Length ³
C3	n	Porosity	Dimensionless

4.10.3.5. Degree of Saturation (TB,PM,,,, TBOPT = DSAT)

Define the degree of saturation as a function of pore pressure (TBPT). Pore-pressure values can be negative only (partially saturated flow), and the degree of saturation lies between 0 and 1. If undefined, the degree of saturation defaults to 1.

Constant	Meaning	Property	Units
C1	p	Pressure	Force/Length ²
C2	s_f	Degree of fluid saturation	None

4.10.3.6. Relative Permeability (TB,PM,,,, TBOPT = RPER)

Define the relative permeability as a function of pore pressure (TBPT). Pore-pressure values can be negative only (partially saturated flow), and relative permeability lies between 0 and 1. If undefined, the relative permeability defaults to 1.

Constant	Meaning	Property	Units
C1	p	Pressure	Force/Length ²
C2	k_r	Relative permeability	None

4.10.3.7. Gravity Magnitude (TB,PM,,,, TBOPT = GRAV)

Define the magnitude of gravity.

Constant	Meaning	Property	Units
C1	g	Gravity magnitude	Force/Length ²

If solid and/or fluid properties are defined (**TB**,PM,,,,SP/FP) and gravity magnitude is defined (**TB**,PM,,,,GRAV), bulk density is calculated as described in [Porous Media Mechanics \(p. 175\)](#).

If gravity magnitude is not defined, it is assumed to be 1.

4.10.3.8. Units of Permeability

The units of permeability are always defined as Length/Time, according to [Darcy's law \(p. 175\)](#). If the specific weight of the fluid γ_f is not defined (**TBOPT** = FP), it is assumed to be 1 (unit = Force/Length³).

4.10.4. Heat-Transfer Properties

Thermal conductivity (**TBOPT** = COND), solid-skeleton specific heat (**TBOPT** = SPHT), and fluid specific heat (**TBOPT** = FLSPHT) are defined via **TB**,THERM. For more information about defining thermal material properties, see [Thermal Properties \(p. 257\)](#).

4.10.5. Thermal-Expansion Material Properties

The solid-skeleton volumetric thermal-expansion coefficient β_s is the sum of the linear thermal-expansion coefficients defined via **TB**,CTE (or **MPTEMP** and **MPDATA**,CTEX/Y/Z).

The fluid volumetric thermal expansion coefficient is directly defined via **TB**,CTE,,,,FLUID.

4.10.6. Transient vs. Static Analysis

A soil-consolidation problem is typically a transient analysis (**ANTYPE**,SOIL or **ANTYPE**,TRANS). A transient analysis considers all rate ($\dot{\epsilon}_v$, $\dot{p}$, $\dot{T}$) and acceleration ($\ddot{u}$) terms in the governing equations ([Equation 4.49 \(p. 176\)](#), [Equation 4.50 \(p. 177\)](#), and [Equation 4.54 \(p. 179\)](#)).

In a geostatic analysis (**ANTYPE**,SOIL with **SSOPT**,GEOSTATIC), all transient terms (acceleration and rate) are ignored.

In a static analysis (**ANTYPE**,STAT), all acceleration terms and all temperature-rate terms ($\dot{T}$) are ignored. However, the program uses a special integration scheme to account for the volumetric strain rate ($\dot{\epsilon}_v$) and pressure rate ($\dot{p}$) terms in the mass balance equation for the pore-fluid phase ([Equation 4.50 \(p. 177\)](#)).

4.10.7. Partially Saturated Porous Media Flow and Coupled-Pore-Pressure-Thermal (CPT) Damping

Soil embankments and dams are examples of semi-saturated porous media. Such structures are often characterized by a phreatic line (free water surface). The soil below the phreatic line is fully saturated, while the soil above is partially saturated. Partially saturated flow results in negative pore pressures. The presence of negative pore pressures is beneficial to the soil. It assures some cohesion in the soil, essential for maintaining its structural integrity.

Partially saturated porous media flow does not introduce any new degrees of freedom to [coupled pore-pressure-thermal elements](#) (CPT nnn); that is, pore pressure is still the extra degree of freedom in addition to the displacement degrees of freedom). Partially saturated behavior is modeled by introducing two new parameters, the degree of saturation and the relative permeability, both of which are functions of the negative pore pressure. The relationships can be defined in tabular form (**TB,PM,,,,DSAT** and **TB,PM,,,,RPER**, respectively).

CPT damping refers to Rayleigh damping, available for CPT nnn elements via the **TB,SDAMP,,,,ALPD/BETD** command.

For more information, see [Example: Initial Degree of Saturation and Relative Permeability in the Advanced Analysis Guide](#).

4.10.8. Additional Resources

For more information about porous media, see:

- The **TB,PM** command documentation
- [Structural-Pore-Fluid-Diffusion-Thermal Analysis in the Coupled-Field Analysis Guide](#)
- [Porous Media Flow in the Theory Reference](#)

4.11. Gasket

The gasket model (**TB,GASKET**) enables simulating the gasket joints with the interface elements. The gasket material is usually under compression and is highly nonlinear. The material also exhibits quite complicated unloading behavior when compression is released.

You can define some general parameters including the initial gap, stable stiffness for numerical stabilization, and stress cap for a gasket in tension. You can also directly input data for the experimentally measured complex pressure closure curves for the gaskets.

Suboptions are also available to define gasket unloading behavior including linear and nonlinear unloading. Linear unloading simplifies the input by defining the starting closure at the compression curves and the slope. Nonlinear unloading option allows you to directly input unloading curves to more accurately model the gasket unloading behavior. When no unloading curves are defined, the material behavior follows the compression curve while it is unloaded.

Enter the general parameters and the pressure closure behavior data via the **TBOPT** option on the **TB,GASKET** command. Input the material data (**TBDATA** or **TBPT**) as shown in the following table:

Gasket Data Type	TBOPT	Constants	Meaning	Input Format
General parameters	PARA	C1	Initial gap. Default = 0 (no initial gap).	TB, GASKET, MATID, , 3, PARA TBDATA, 1, C1, C2, C3
		C2	Scaling factor to produce stable stiffness $K_s = C2 * K_0$, where $K_0 = Y1 / X1$ is the initial compressive loading stiffness. Default = 1E-7. [1]	

Gasket Data Type	TBOPT	Constants	Meaning	Input Format
		C3	Maximum tension stress allowed when the gasket material is in tension. Default = 0 (no tension stress in the gasket material).	
Compression load closure curve	COMP	Xi	Closure value.	TB, GASKET, MATID, , 2, COMP TBPT, , X1, Y1 TBPT, , X2, Y2
		Yi	Pressure value.	
Linear unloading data	LUNL	Xi	Closure value on compression curve where unloading started.	TB, GASKET, MATID, , 2, LUNL TBPT, , X1, Y1 TBPT, , X2, Y2
		Yi	Unloading slope value.	
Nonlinear unloading data [2]	NUNL	Xi	Closure value.	TB, GASKET, MATID, , 2, NUNL TBPT, , X1, Y1 TBPT, , X2, Y2
		Yi	Pressure value.	
Transverse shear stiffness and membrane stiffness per unit thickness (Unit = Force / Length ³)	TSS	XY, XZ	Transverse shear stiffness. Default = Maximum of stable stiffness (C2) or 1E - 4 * K0 (where K0 = Y1 / X1 is the initial compressive loading stiffness). [3] TSS XZ is set to TSS XY if not specified.	TB, GASKET, , , 2, TSS TBDATA, 1, TSSXY, TSSXZ, MSTIF
		MS	Membrane stiffness. TSS MS is set to TSS XY + TSS XZ if not specified.	
Transverse shear stiffness and membrane stiffness (Unit = Force / Length ²) [4]	TSMS	GXY	Shear modulus XY.	TB, GASKET, MATID, , 6, TSMS TBDATA, , GXY, GXZ, EY, EZ, GYZ, NUYZ
		GXZ	Shear modulus XZ.	
		EYY	Elastic modulus YY.	
		EZZ	Elastic modulus ZZ.	
		GYZ	Shear modulus YZ.	
		NUYZ	Minor Poisson's ratio YZ.	

1. Stable stiffness is used for numerical stabilization. For example, numerical instability can occur when the gasket is opened up, thus contributing no stiffness to the element nodes. A realistic value of C2 can be given by $C2 = (E / h) / K0$, where E is the Young's modulus characteristic of the gasket material, h is the initial gasket thickness and K0 is the initial gasket compression stiffness. As the stable stiffness is adopted solely for numerical stabilization, however, a proper value of C2 should be $C2 = C * (E / h) / K0$, where C is a small scaling factor (such as 1E-7). Default: C2 = 1E-7.
2. Multiple curves may be required to define the complex nonlinear unloading behavior of a gasket material.

When several nonlinear unloading curves are defined, the starting point of each unloading curve must be on the compression curve to ensure that the gasket unloading behavior is correctly simulated.

The temperature-dependency of unloading data does not need to match the compression data; however, if there is a missing temperature, the program uses linear interpolation to obtain the material data of the missing temperature, possibly resulting in a mismatch between the compression data and the unloading data. It is therefore good practice to have the same number of temperatures and temperature points for each unloading curve and compression curve.

3. With a known shear modulus G characteristic of the gasket material, the transverse shear stiffness values TSS_{xy} and TSS_{xz} can be calculated via $TSS_{xy} = TSS_{xz} = G / h$, where h is the initial gasket thickness (Unit = Force / Length³). If using transverse shear stiffness primarily for overcoming numerical instability, the values should be further scaled with a small number C , as shown in this example:

$$TSS_{xy} = TSS_{xz} = C * (G / h), \text{ where } C = 1E-4.$$

4. If specified, $TBOPT = TSMS$ takes priority over $TBOPT = TSS$.

GXY defaults to maximum of initial compression stiffness * factor (1E-4) or stable stiffness ($C2$) (defined via **TB,GASKET,,,PARA**). Initial compression stiffness is defined as $h * Y1/X1$, where $(X1,Y1)$ is the first data point provided in the gasket-loading curve.

If not specified, Gxz defaults to GXY , EYY defaults to $GXY + Gxz$, and EZZ defaults to EYY . (If EZZ is specified, GYZ and $NUYZ$ are required.)

$NUYZ$ defaults to 0.0.

Units of the moduli parameters are Force / Length². Poisson's ratio parameter $NUYZ$ is unitless.

You can enter temperature-dependent data (**TBTEMP**) for any of the gasket data types. For the first temperature curve, issue **TB,GASKET,,,TBOPT** (where $TBOPT$ is the gasket material option), then input the first temperature (**TBTEMP**), followed by the data (**TBDATA** or **TBPT** depending on the $TBOPT$, as shown in the table). The number of data points defined (**TBPT** commands issued) for the specified $TBOPT$ must be the same across all temperatures.

The program automatically interpolates the temperature data to the material points using linear interpolation. When the temperature is out of the specified range, the closest temperature point is used.

For more information, see [Gasket Material](#) in the *Mechanical APDL Theory Reference*.

For a detailed description of the gasket joint simulation capability, see [Gasket Joints Simulation](#) in the *Structural Analysis Guide*.

4.12. Swelling

Swelling (**TB,SWELL**) is a material enlargement (volume expansion) caused by neutron bombardment or other effects (such as moisture). The swelling strain rate is generally nonlinear and is a function of factors such as temperature, time, neutron flux level, stress, and moisture content.

Irradiation-induced swelling and creep apply to metal alloys that are exposed to nuclear radiation. However, the swelling equations and the fluence input may be completely unrelated to nuclear swelling. You can also model other types of swelling behavior, such as moisture-induced volume expansion.

Swelling strain is modeled using additive decomposition of strains, expressed as:

$$\dot{\epsilon} = \dot{\epsilon}_{el} + \dot{\epsilon}_{pl} + \dot{\epsilon}_{cr} + \dot{\epsilon}_{th} + \dot{\epsilon}_{sw}$$

where ϵ is the total strain, ϵ_{el} is the elastic strain, ϵ_{pl} is the plastic strain, ϵ_{cr} is the creep strain, ϵ_{th} is the thermal strain, and ϵ_{sw} is the swelling strain. The mechanical strain is the total strain minus the thermal and swelling strains. Swelling strain is assumed to be isotropic. You can combine swelling strain with other material models such as plasticity and creep; however, you cannot use swelling with any hyperelasticity or anisotropic hyperelasticity material model.

Irradiation-induced swelling is generally accompanied by irradiation creep for metals and composites, such as silicon carbide (SiC). The irradiation-induced swelling strain rate may depend on temperature, time, fluence (the flux x time), and stress, such as:

$$\dot{\epsilon}_{sw} = \epsilon_{sw}(t, T, \Phi_t, \sigma)$$

where t is time, T is the temperature, Φ_t is the fluence, and σ is the stress. Temperatures used in the swelling equations should be based on an absolute scale (**TOFFST**). Specify temperature and fluence values via the **BF** or **BFE** command.

The following options for modeling swelling are available:

- *Linear swelling* defines swelling strain rate as a function of fluence rate, expressed as:

$$\dot{\epsilon}_{sw} = C \dot{\Phi}_t$$

where C is the swelling constant, which may depend on temperature.

- *Exponential swelling* defines swelling strain as a function of fluence, expressed as:

$$\epsilon_{sw} = C_1 + C_2 \Phi_t + C_3 [\exp(-C_4 \Phi_t) - 1]$$

- A *user-defined swelling* option is available if you wish to create your own swelling function. For more information, see [userswstrain](#) in the *Programmer's Reference*.

Swelling equations are material-specific and are empirical in nature.

For highly nonlinear swelling strain vs. fluence curves, it is good practice to use a small fluence step for better accuracy and solution stability. If time is changing, a constant flux requires a linearly changing fluence (because the swelling model uses fluence $[\Phi_t]$ rather than flux $[\Phi]$).

Initialize the swelling table (**TB,SWELL**) with the desired data-table option (**TBOPT**), as follows:

Swelling Model Options (TB,SWELL,,TBOPT)			
Option (TBOPT)	Constant	Description	Constant Value Input
LINE	C1	Linear swelling	TB DATA,1,C1
EXPT	C1, C2, C3, C4	Exponential swelling	TB DATA,1,C1,C2,C3,C4

Swelling Model Options (TB,SWELL,,TBOPT)			
USER	C1, ..., C _n	User-defined	TBDATA,1,C1,C2,...

Issue the **TBDATA** command to enter the swelling table constants (up to six per command), as shown in the table.

For a list of the elements that you can use with the swelling model, see [Material Model Element Support](#) (p. 5). The swelling model does not support sections (**SECTYPE**).

For more information about this material model, see [Swelling](#) in the *Structural Analysis Guide*.

4.13. Shape Memory Alloy (SMA)

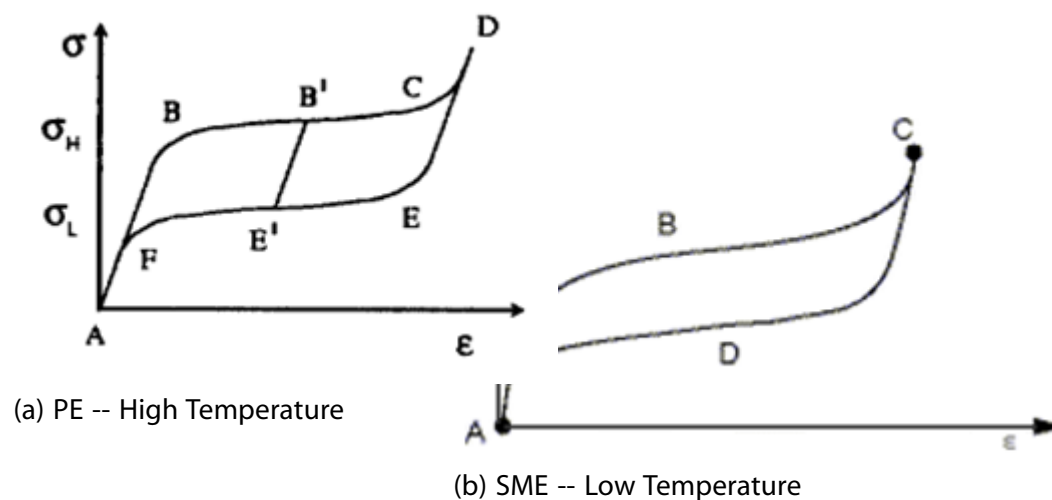
A shape memory alloy (SMA) is a metallic alloy that returns to its original shape after deformation. Upon loading and unloading cycles, an SMA can undergo large deformation without showing residual strains (pseudoelasticity effect, also often called superelasticity), and can recover its original shape through thermal cycles (the shape memory effect).

Such distinct material behavior is due to the material microstructure in which there exists two different crystallographic structures, one characterized by austenite (A), and another one by martensite (M). Austenite is the crystallographically more-ordered phase, and martensite is the crystallographically less-ordered phase. The key characteristic of an SMA is the occurrence of a martensitic phase transformation.

Typically, the austenite is stable at high temperatures and low stress, while the martensite is stable at low temperatures and high stress. The reversible martensitic phase transformation results in unique effects: the pseudoelasticity (PE) and the shape memory effect (SME).

As shown by (a) in the following figure, whenever σ_L is positive, the specimen recovers its original shape completely and returns to a stress-free configuration (PE).

Figure 4.50: Pseudoelasticity (PE) and Shape Memory Effect (SME)



As shown by (b) in the figure, when σ_L is negative, residual strains (E and E') can be observed after unloading into a stress-free configuration. If the material is heated, then eventually σ_L becomes positive; however, the admissible configuration under a stress-free state points to A. The material therefore undergoes an inverse transformation process (SME).

Nitinol

A typical shape memory alloy is Nitinol, a nickel titanium (Ni-Ti) alloy discovered in the 1960s at the U.S. Naval Ordnance Laboratory (NOL). The acronym NiTi-NOL (or Nitinol) has since been commonly used when referring to Ni-Ti-based shape memory alloys.

Two SMA material model options (accessed via **TB,SMA**) are available, one for simulating [superelastic behavior](#) (p. 189) and the other for simulating the [shape memory effect behavior](#) (p. 193) of shape memory alloys.

The material option for superelasticity is based on Auricchio *et al.* [1] in which the material undergoes large-deformation without showing permanent deformation under isothermal conditions, as shown by (a) in [Figure 4.50: Pseudoelasticity \(PE\) and Shape Memory Effect \(SME\)](#) (p. 188). The material option for the shape memory effect is based on the 3-D thermomechanical model for stress-induced solid phase transformations [2] [3] [4].

The following shape memory alloy topics are available:

- [4.13.1.SMA Model for Superelasticity](#)
- [4.13.2.SMA Material Model with Shape Memory Effect](#)
- [4.13.3.Result Output of Solution Variables](#)
- [4.13.4.Element Support for SMA](#)
- [4.13.5.Learning More About Shape Memory Alloy](#)

4.13.1. SMA Model for Superelasticity

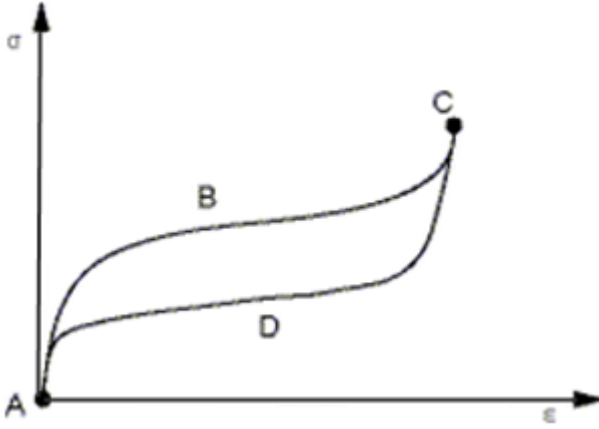
The following topics are available for the SMA superelasticity option:

- [4.13.1.1.Constitutive Model for Superelasticity](#)
- [4.13.1.2.Material Parameters for the Superelastic SMA Material Model](#)

4.13.1.1. Constitutive Model for Superelasticity

From a macroscopic perspective, the phase-transformation mechanisms involved in superelastic behavior are:

1. Austenite to martensite (A->S)
2. Martensite to austenite (S->A)
3. Martensite reorientation (S->S)

Figure 4.51: Typical Superelasticity Behavior

Two of the phase transformations are considered here: $A \rightarrow S$ and $S \rightarrow A$. The material is composed of two phases, the austenite (A) and the martensite (S). Two internal variables, the martensite fraction (ξ_S) and the austenite fraction (ξ_A), are introduced. One of them is a dependent variable, and they are assumed to satisfy the relation expressed as:

$$\xi_S + \xi_A = 1$$

The independent internal variable chosen here is ξ_S .

The material behavior is assumed to be isotropic. The pressure dependency of the phase transformation is modeled by introducing the Drucker-Prager loading function, as follows:

$$F = q + 3\alpha p$$

$$q = \sqrt{\frac{3}{2}} S:S$$

$$S = \sigma - p1$$

$$p = \frac{1}{3} \sigma:1$$

where α is the material parameter, σ is the stress, and 1 is the identity tensor.

The evolution of the martensite fraction, ξ_S , is then defined as follows:

$$\dot{\xi}_S = \begin{cases} -H^{AS}(1-\xi_S) \frac{\dot{F}}{F-R_f^{AS}} & A \rightarrow S \text{ transformation} \\ H^{SA} \xi_S \frac{\dot{F}}{F-R_f^{SA}} & S \rightarrow A \text{ transformation} \end{cases}$$

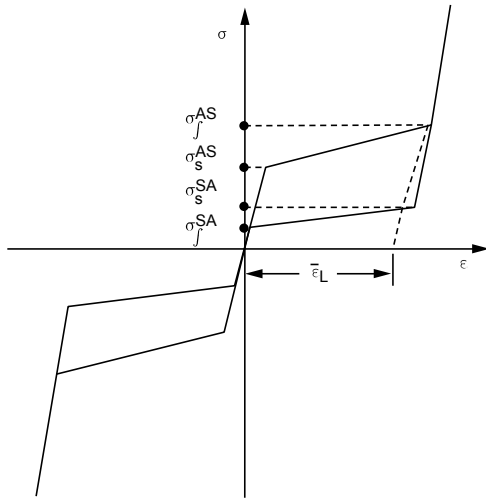
where:

$$R_f^{AS} = \sigma_f^{AS}(1+\alpha)$$

$$R_f^{SA} = \sigma_f^{SA}(1+\alpha)$$

where σ_f^{AS} and σ_f^{SA} are the material parameters shown in the following figure:

Figure 4.52: Idealized Stress-Strain Diagram of Superelastic Behavior



$$H^{AS} = \begin{cases} 1 & \text{if } \begin{cases} R_s^{AS} < F < R_f^{AS} \\ \dot{F} > 0 \end{cases} \\ 0 & \text{otherwise} \end{cases}$$

$$H^{SA} = \begin{cases} 1 & \text{if } \begin{cases} R_f^{SA} < F < R_s^{SA} \\ \dot{F} < 0 \end{cases} \\ 0 & \text{otherwise} \end{cases}$$

$$R_s^{AS} = \sigma_s^{AS} (1 + \alpha)$$

$$\sigma_s^{AS} \quad \text{and} \quad \sigma_s^{SA}$$

where σ_s^{AS} and σ_s^{SA} are the material parameters shown in [Figure 4.52: Idealized Stress-Strain Diagram of Superelastic Behavior \(p. 191\)](#).

The material parameter α characterizes the material response in tension and compression. If tensile and compressive behaviors are the same, then $\alpha = 0$. For a uniaxial tension-compression test, α can be related to the initial value of austenite to martensite phase transformation in tension and compression (σ_c^{AS} and σ_t^{AS} , respectively) as:

$$\alpha = \frac{\sigma_c^{AS} - \sigma_t^{AS}}{\sigma_c^{AS} + \sigma_t^{AS}}$$

The stress-strain relation is:

$$\sigma = \mathbf{D} : (\varepsilon - \varepsilon_{tr})$$

$$\dot{\varepsilon}_{tr} = \dot{\xi} \bar{\varepsilon}_L \frac{\partial F}{\partial \sigma}$$

where $\mathbf{D}$ is the elastic stiffness tensor, ε_{tr} is the transformation strain tensor, and $\bar{\varepsilon}_L$ is the material parameter shown in [Figure 4.52: Idealized Stress-Strain Diagram of Superelastic Behavior \(p. 191\)](#).

To model the difference between the austenite and martensite elastic properties, the elastic modulus E (used in the material constitutive model) is a function of the martensite fraction ξ_S , the elastic modulus of pure austenite E_A (the initial input elastic modulus), and the elastic modulus of pure martensite E_S :

$$E = \frac{E_A}{1 + \left(\frac{E_A}{E_S} - 1\right)\xi_S}$$

4.13.1.2. Material Parameters for the Superelastic SMA Material Model

To model the superelastic behavior of shape memory alloys, initialize the data table via the **TB,SMA** command's **SUPE** option.

Define the elastic behavior in the austenite state (**MP**).

The superelastic SMA option is described by [six constants](#) (p. 192) that define the stress-strain behavior in loading and unloading for the uniaxial stress-state.

For each data set, define the temperature (**TBTEMP**), then define [constants C1 through C6](#) (p. 192) (**TBDATA**). You can define up to 99 sets of temperature-dependent constants in this manner.

Table 4.29: Superelastic Option Constants

Constant	Meaning	Property
C1	σ_s^{AS}	Starting stress value for the forward phase transformation
C2	σ_f^{AS}	Final stress value for the forward phase transformation
C3	σ_s^{SA}	Starting stress value for the reverse phase transformation
C4	σ_f^{SA}	Final stress value for the reverse phase transformation
C5	$\bar{\epsilon}_L$	Maximum residual strain
C6	α	Parameter measuring the difference between material responses in tension and compression
C7	E_S	Elastic modulus of the full martensite phase If 0 or undefined, the martensite and austenite phases share the same elastic modulus $E = E_A = E_S$

Example 4.34: Defining Elastic Properties of the Austenite Phase

```
MP,EX,1,60000.0
MP,NUXY,1,0.36
Define SMA material properties
TB,SMA,1,,,SUPE
TBDATA,1, 520, 600, 300, 200, 0.07, 0.0
```

4.13.2. SMA Material Model with Shape Memory Effect

The following topics concerning SMA and the shape memory effect are available:

[4.13.2.1. The Constitutive Model for Shape Memory Effect](#)

[4.13.2.2. Material Parameters for the Shape Memory Effect Option](#)

4.13.2.1. The Constitutive Model for Shape Memory Effect

The shape memory effect was based on a 3-D thermomechanical model for stress-induced solid phase transformations that was presented in [2] [3][4]. Within the framework of classical irreversible thermodynamics, the model is able to reproduce all of the primary features relative to shape memory materials in a 3-D stress state. The free energy potential is set to:

$$\Psi(\varepsilon, T, \varepsilon_{tr}) = \frac{1}{2}(\varepsilon - \varepsilon_{tr}) : D : (\varepsilon - \varepsilon_{tr}) + \tau_M(T) \|\varepsilon'_{tr}\| + \frac{1}{2} h \|\varepsilon_{tr}\|^2 + I_{\varepsilon'_{tr}}(\varepsilon'_{tr})$$

where:

$\mathbf{D}$ = material elastic stiffness tensor

ε = total strain

ε_{tr} = total transformation strain

ε'_{tr} = deviatoric transformation strain

$\tau_M(T)$ = a positive and monotonically increasing function of the temperature as $\langle \beta(T - T_0) \rangle^+$ in which $\langle \cdot \rangle^+$ is the positive part of the argument (also known as Maxwell stress).

β = material parameter

T = temperature

T_0 = temperature below which no austenite is observed in a stress-free state

h = material parameter related to the hardening of the material during the phase transformation

$I_{\varepsilon'_{tr}}(\varepsilon'_{tr})$ = indicator function introduced to satisfy the constraint on the transformation norm [1] in which

$$I_{\varepsilon'_{tr}}(\varepsilon'_{tr}) = \begin{cases} 0 & 0 \leq \|\varepsilon'_{tr}\| \leq \bar{\varepsilon}_L \\ +\infty & \text{otherwise} \end{cases}$$

from which we have

$$\sigma = \frac{\partial \Psi}{\partial \varepsilon}$$

$$\mathbf{X}_{tr} \in -\frac{\partial \Psi}{\partial \varepsilon'_{tr}}$$

where $\mathbf{X}_{tr}$ is defined as the transformation stress.

Stresses, strains, and the transformation strains are then related as follows:

$$\sigma = \mathbf{D} : (\varepsilon - \varepsilon_{tr})$$

Splitting the stress into deviatoric and volumetric components, we have

$$\mathbf{S} = \boldsymbol{\sigma} - p\mathbf{1}$$

$$p = \frac{1}{3}\boldsymbol{\sigma} : \mathbf{1}$$

where $\mathbf{S}$ is the deviatoric stress and p is the volumetric stress (also called hydrostatic pressure)

The transformation stress is given as follows:

$$\mathbf{X}_{tr} = \mathbf{S} - \left[\tau_M(T) + h\|\boldsymbol{\varepsilon}_{tr}\| + \gamma \right] \frac{\boldsymbol{\varepsilon}_{tr}}{\|\boldsymbol{\varepsilon}_{tr}\|}$$

where γ is defined by

$$\gamma = \begin{cases} 0 & 0 \leq \|\boldsymbol{\varepsilon}_{tr}\| < \bar{\varepsilon}_L \\ \geq 0 & \|\boldsymbol{\varepsilon}_{tr}\| = \bar{\varepsilon}_L \end{cases}$$

where $\bar{\varepsilon}_L$ is a maximum transformation strain.

Numerous experimental tests show an asymmetric behavior of SMA in tension and compression, and suggest describing SMA as an isotropic material with a Prager-Lode-type limit surface. Accordingly, the following yield function is assumed:

$$F(\mathbf{X}_{tr}) = \sqrt{2J_2} + m \frac{J_3}{J_2} - R$$

where $\mathbf{X}_{tr}$ is the transformation stress, J_2 and J_3 are the second and third invariants of transformation stress, m is a material parameter related to Lode dependency, and R is the elastic domain radius.

J_2 and J_3 are defined as follows:

$$J_2 = \frac{1}{2}(\mathbf{X}_{tr}^2 : \mathbf{1})$$

$$J_3 = \frac{1}{3}(\mathbf{X}_{tr}^3 : \mathbf{1})$$

The evolution of transformation strain is defined as:

$$\dot{\boldsymbol{\varepsilon}}_{tr} = \dot{\xi} \bar{\varepsilon}_L \frac{\partial F}{\partial \boldsymbol{\sigma}}$$

where ξ is an internal variable and is called as transformation strain multiplier. ξ and $\mathbf{F}(\mathbf{X}_{tr})$ must satisfy the classical Kuhn-Tucker conditions, as follows:

$$\begin{aligned} \dot{\xi} &\geq 0, \\ \dot{\xi} F(\mathbf{X}_{tr}) &= 0 \end{aligned}$$

which also reduces the problem to a constrained optimization problem.

The shape memory effect option is described by [seven constants \(p. 196\)](#) that define the stress-strain behavior of material in loading and unloading cycles for the uniaxial stress-state and thermal loading.

For each data set, define the temperature (**TBTEMP**), then define constants [C1 through C7 \(p. 196\)](#) (**TBDATA**). You can define up to 99 sets of temperature-dependent constants in this manner.

Table 4.30: Shape Memory Effect Option Constants

Constant	Meaning	Property
C1	h	Hardening parameter
C2	T_o	Reference temperature
C3	R	Elastic limit
C4	β	Temperature scaling parameter
C5	$\bar{\epsilon}_L$	Maximum transformation strain
C6	E_m	Martensite modulus
C7	m	Lode (p. 194) dependency parameter

Example 4.35: Defining Shape Memory Effect Properties of the Austenite Phase

```
MP,EX,1,60000.0
MP,NUXY,1,0.36
Define SMA material properties
TB,SMA,1,,,MEFF
TBDATA,1,1000, 223, 50, 2.1, 0.04, 45000
TBDATA,7,0.05
```

4.13.3. Result Output of Solution Variables

For postprocessing, solution output is as follows:

- Stresses are output as S.
- Elastic strains are output as EPEL
- Transformation strains, ϵ_{tr} , are output as plastic strain EPPL
- The ratio of the equivalent transformation strain to maximum transformation strain, $\|\epsilon_{tr}\|/\bar{\epsilon}_L$, is available as part of nonlinear solution record NL, and can be processed as component EPEQ of NL.
- Elastic strain energy density is available as part of the strain energy density record SEND (ELASTIC).

4.13.4. Element Support for SMA

Support for SMA material models with the superplasticity option (**TB,SMA,,,,SUPE**) is available with [current-technology](#) plane, solid, and solid-shell elements where 3-D stress states are applicable (including 3-D solid elements, solid-shell elements, 2-D plane strain, axisymmetric elements, and solid pipe elements).

Support for SMA material models with the memory-effect option (**TB,SMA,,,,MEFF**) is available with current-technology beam, shell, plane, solid, and solid-shell elements (including 3-D solid elements, solid-shell elements, 2-D plane stress and strain, axisymmetric elements, and solid pipe elements).

For specific element support for SMA, see [Material Model Element Support](#) (p. 5).

4.13.5. Learning More About Shape Memory Alloy

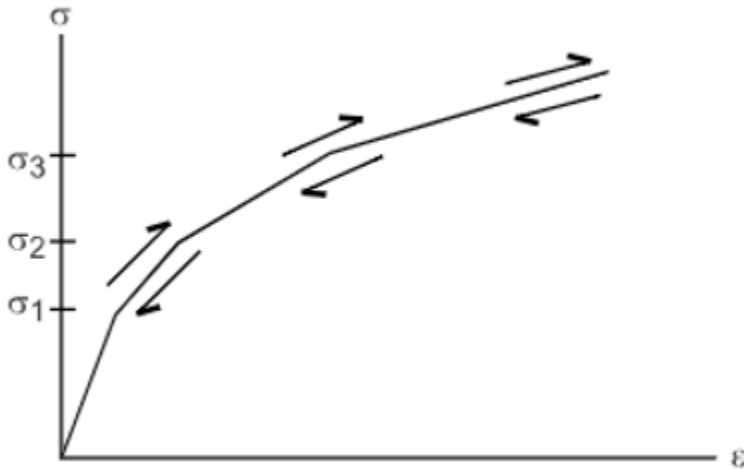
A considerable body of literature exists concerning shape memory alloy material models. The following list of resources offers a wealth of information but is by no means exhaustive:

1. Auricchio, F. "A Robust Integration-Algorithm for a Finite-Strain Shape-Memory-Alloy." *International Journal of Plasticity*. 17 (2001): 971-990.
2. Souza, A. C., E. N. Mamiya, N. Zouain. "Three-Dimensional Model for Solids Undergoing Stress-Induced Phase Transformations." *European Journal of Mechanics-A/Solids* . 17 (1998): 789-806.
3. Auricchio, F., R. L. Taylor, J. Lubliner. "Shape-Memory Alloys: Macromodeling and Numerical Simulations of the Superelastic Behavior." *Computational Methods in Applied Mechanical Engineering*. 146, 1 (1997): 281-312.
4. Auricchio, F., L. Petrini. "Improvements and Algorithmical Considerations on a Recent Three-Dimensional Model Describing Stress-Induced Solid Phase Transformations." *International Journal for Numerical Methods in Engineering*. 55 (2005): 1255-1284.
5. Auricchio, F., D. Fugazza, R. DesRoches. "Numerical and Experimental Evaluation of the Damping Properties of Shape-Memory Alloys." *Journal of Engineering Materials and Technology*. 128:3 (2006): 312-319.

For an example analysis, see [Shape Memory Alloy \(SMA\) with Thermal Effect](#) in the [Technology Demonstration Guide](#).

4.14. Multilinear Elasticity

The multilinear elasticity material model causes unloading to occur along the same path as loading. This behavior is conservative (path-independent). The plastic strain should be interpreted as a *pseudo-plastic strain*, as it returns to zero when the material is unloaded.

Figure 4.54: Stress-Strain Behavior for Multilinear Elasticity

Multilinear material behavior is described by a piece-wise linear stress-strain curve, starting at the origin, with positive stress and strain values. The curve is continuous from the origin through a set of designated stress-strain points. Successive slopes can be greater than the preceding slope; however, no slope can be greater than the elastic modulus of the material. The slope of the first curve segment usually corresponds to the elastic modulus of the material, although the elastic modulus can be input as greater than the first slope to ensure that all slopes are $\leq$ the elastic modulus.

Define the elastic behavior via **MP** or **TB,ELASTIC**. Define a material data table for the stress-strain values via **TB,MELAS**, and specify the values via **TBPT**.

TBPT Values for Stress-Strain		
Constant	Units	Property
X	Dimensionless	Strain value
Y	Force/Area	Corresponding stress value

Example 4.36: Defining Stress-Strain Values for Elastic Behavior

```

TB, ELASTIC,1,2
TBFIELD, TEMP,70
TBDATA, ,1E4,0.3
TBFIELD, TEMP,200
TBDATA, , 1E3,0.3

TB, MELAS,1,2,3
TBFIELD, TEMP, 70
TBPT, ,0.00005,0.5
TBPT, ,0.00105,0.8
TBPT, ,0.00305,1.1
TBFIELD, TEMP, 200
TBPT, ,0.0010,1.0
TBPT, ,0.0020,1.2
TBPT, ,0.0040,1.4

```

The program uses the total strain components ϵ_n to calculate an equivalent total strain measure:

$$\epsilon_e^t = \frac{1}{\sqrt{2(1+\nu)}} \left[(\epsilon_x - \epsilon_y)^2 + (\epsilon_y - \epsilon_z)^2 + (\epsilon_z - \epsilon_x)^2 + \frac{3}{2}(\epsilon_{xy})^2 + \frac{3}{2}(\epsilon_{yz})^2 + \frac{3}{2}(\epsilon_{xz})^2 \right]^{\frac{1}{2}}$$

which is used with the input stress-strain curve to get an equivalent value of stress σ_e .

The elastic (linear) component of strain can then be calculated:

$$\boldsymbol{\varepsilon}_n^{el} = \frac{\sigma_e}{E \varepsilon_e^t} \boldsymbol{\varepsilon}_n$$

and the nonlinear portion is:

$$\boldsymbol{\varepsilon}_n^{pl} = \boldsymbol{\varepsilon}_n - \boldsymbol{\varepsilon}_n^{el}$$

To avoid an unsymmetric matrix, only the symmetric portion of the tangent stress-strain matrix is used:

$$\mathbf{D}_{ep} = \frac{\sigma_e}{E \varepsilon_e} \mathbf{D}$$

which is the secant stress-strain matrix.

Also see [Material Model Element Support \(p. 5\)](#) for this material model.

4.15. MPC184 Joint

The **TB,JOIN** option allows you to impose linear and nonlinear elastic stiffness and damping behavior or Coulomb friction behavior on the available components of relative motion of an **MPC184** joint element. The stiffness and damping behaviors described here apply to all joint elements except the **weld**, **orient**, and **spherical** joints. The Coulomb friction behavior described here applies only to the **revolute**, **slot**, and **translational** joints.

The **TB** command may be repeated with the same material ID number to specify both the stiffness and damping behavior.

The following joint material models are available:

[4.15.1. Linear Elastic Stiffness and Damping Behavior](#)

[4.15.2. Nonlinear Elastic Stiffness and Damping Behavior](#)

[4.15.3. Frictional Behavior](#)

4.15.1. Linear Elastic Stiffness and Damping Behavior

Input the linear stiffness or damping behavior for the relevant components of relative motion of a joint element by specifying the terms as part of a 6 x 6 matrix with data-table commands as described below.

The 6 x 6 matrix for linear stiffness or damping behavior is as follows:

$$\begin{bmatrix} D_{11} & & & & & \\ D_{21} & D_{22} & & & & \\ D_{31} & D_{32} & D_{33} & & & \\ D_{41} & D_{42} & D_{43} & D_{44} & & \\ D_{51} & D_{52} & D_{53} & D_{54} & D_{55} & \\ D_{61} & D_{62} & D_{63} & D_{64} & D_{65} & D_{66} \end{bmatrix}$$

Enter the stiffness or damping coefficient of the matrix in the data table with **TB** set of commands. Initialize the constant table with **TB,JOIN,,,STIF** (for stiffness behavior) or **TB,JOIN,,,DAMP** (for damping behavior). Define the temperature with **TBTEMP**, followed by the relevant constants input with **TBDATA** commands. Matrix terms are linearly interpolated between temperature points. Based on the joint type, the relevant constant specification is as follows:

Joint Element	Constant	Meaning
x-axis Revolute joint	C16	Term D_{44}
z-axis Revolute joint	C21	Term D_{66}
Universal joint	C16, C18, C21	Terms D_{44} , D_{64} , D_{66}
Slot joint	C1	Term D_{11}
Point-in-plane joint	C7, C8, C12	Terms D_{22} , D_{32} , D_{33}
Translational joint	C1	Term D_{11}
x-axis Cylindrical joint	C1, C4, C16	Terms D_{11} , D_{41} , D_{44}
z-axis Cylindrical joint	C12, C15, C21	Terms D_{33} , D_{63} , D_{66}
x-axis Planar joint	C7, C8, C9, C12, C13, C16	Terms D_{22} , D_{32} , D_{42} , D_{33} , D_{43} , D_{44}
z-axis Planar joint	C1, C2, C6, C7, C11, C21	Terms D_{11} , D_{21} , D_{61} , D_{22} , D_{62} , D_{66}
General joint	Use appropriate entries based on unconstrained degrees of freedom.	---
Screw joint	C12, C15, C21	Terms D_{33} , D_{63} , D_{66}

The following example shows how you would define the uncoupled linear elastic stiffness behavior for a universal joint at the two available components of relative motion, with two temperature points:

```
TB,JOIN,1,2,,STIF ! Activate JOIN material model with linear elastic stiffness
TBTEMP,100.0 ! Define first temperature
TBDATA,16,D44 ! Define constant D44 in the local ROTX direction
TBDATA,21,D66 ! Define constant D66 in the local ROTZ direction
TBTEMP,200.0 ! Define second temperature
TBDATA,16,D44 ! Define constant D44 in the local ROTX direction.
TBDATA,21,D66 ! Define constant D66 in the local ROTZ direction.
```

4.15.2. Nonlinear Elastic Stiffness and Damping Behavior

You can specify nonlinear elastic stiffness as a displacement (rotation) versus force (moment) curve using the **TB,JOIN** command with a suitable **TBOPT** setting.

Use the **TBPT** command to specify the data points or [specify the name of a function \(p. 202\)](#) that defines the curve on the **TB** command. (Use the [Function Tool](#) to generate the specified function.) The values may be temperature-dependent.

You can specify nonlinear damping behavior in a similar manner by supplying velocity versus damping force (or moment).

The appropriate **TBOPT** labels for each joint element type are shown in the following tables. For a description of each **TBOPT** label, see ["JOIN -- Joint Element Specifications"](#) in the **TB** command documentation.

Nonlinear Stiffness Behavior	
Joint Element	TBOPT on TB command
x-axis Revolute joint	JNSA, JNS4
z-axis Revolute joint	JNSA, JNS6
Universal joint	JNSA, JNS4, and JNS6
Slot joint	JNSA and JNS1
Point-in-plane joint	JNSA, JNS2, and JNS3
Translational joint	JNSA and JNS1
x-axis Cylindrical joint	JNSA, JNS1, and JNS4
z-axis Cylindrical joint	JNSA, JNS3, and JNS6
x-axis Planar joint	JNSA, JNS2, JNS3, and JNS4
z-axis Planar joint	JNSA, JNS1, JNS2, and JNS6
General joint	Use appropriate entries based on unconstrained degrees of freedom
Screw joint	JNSA, JNS3, and JNS6

Nonlinear Damping Behavior	
Joint Element	TBOPT on TB command
x-axis Revolute joint	JNDA, JND4
z-axis Revolute joint	JNDA, JND6
Universal joint	JNDA, JND4, and JND6
Slot joint	JNDA and JND1
Point-in-plane joint	JNDA, JND2, and JND3
Translational joint	JNDA and JND1
x-axis Cylindrical joint	JNDA, JND1, and JND4
z-axis Cylindrical joint	JNDA, JND3, and JND6
x-axis Planar joint	JNDA, JND2, JND3, and JND4
z-axis Planar joint	JNDA, JND1, JND2, and JND6
General joint	Use appropriate entries based on unconstrained degrees of freedom
Screw joint	JNDA, JND3, and JND6

The following example illustrates the specification of nonlinear stiffness behavior for a revolute joint that has only one available component of relative motion (the rotation around the axis of revolution). Two temperature points are specified.

```

TB, JOIN, 1, 2, 2, JNS4
TBTEMP, 100.
TBPT, , rotation_value_1, moment_value_1
TBPT, , rotation_value_2, moment_value_2
TBTEMP, 200.0
TBPT, , rotation_value_1, moment_value_1
TBPT, , rotation_value_2, moment_value_2

```

4.15.2.1. Specifying a Function Describing Nonlinear Stiffness Behavior

When specifying a function that describes the nonlinear stiffness behavior, the **Function Tool** allows the force to be defined as a function of temperature and relative displacement; the two independent variables are named as TEMP and DJU. Similarly, when specifying a function that describes the nonlinear damping behavior, the **Function Tool** allows the damping force to be defined as a function of temperature and relative velocity; the two independent variables are identified as TEMP and DJV.

Example Consider a function where the damping force varies with temperature and relative velocity:

$$F = (-0.005 * \text{Temperature} + 0.25) * \text{Relative Velocity}$$

Define the function using the **Function Editor**, then retrieve and load it using the **Function Loader**. (The editor and the loader are both components of the **Function Tool**.)

Assuming a function name of dampfunc, you can then use the **TB** command to define the joint material:

```
TB, JOIN, 1, , , JND4, , %dampfunc%
```

For more information about the **Function Tool** utility, see [Using the Function Tool](#) in the *Basic Analysis Guide*.

4.15.3. Frictional Behavior

Frictional behavior along the unrestrained components of relative motion influences the overall behavior of the Joints. You can model Coulomb friction for joint elements via the **TB,JOIN** command with an appropriate **TBOPT** label. The joint frictional behavior can be specified only for the following joints: [Revolute joint](#), [Slot joint](#), and [Translational joint](#).

The friction parameters are described below.

Coulomb Friction Coefficient Specification

There are three options for defining the Coulomb friction coefficient.

- Define a single value of the Coulomb friction coefficient by specifying **TBOPT = MUSx**, where the value of x depends on the joint under consideration. Use the **TBDATA** command to specify the value of the friction coefficient.
- Define the Coulomb friction coefficient as a function of the sliding velocity. Use **TBOPT = MUSx** (as stated above) and use the **TBPT** command to specify the data values.
- Use the exponential law for friction behavior. Specify **TBOPT = EXPx**, where the value of x depends on the joint under consideration, and use the **TBDATA** command to specify the values required for the exponential law. In this case, the **TBDATA** command format is:

TBDATA, μ_s , μ_d , c

where μ_s is the coefficient of friction in the static regime, μ_d is the coefficient of friction in the dynamic regime, and c is the decay coefficient.

Maximum or Critical Force/Moment

- The maximum allowable value of critical force/moment can be specified using $TBOPT = TMXx$, where x depends on the joint under consideration.

Elastic Slip

- The elastic slip can be specified by setting $TBOPT = SLx$, where x depends on the joint under consideration.
- If the stick-stiffness value is not specified, then this value along with the critical force/moment is used to determine the stick-stiffness.
- If the elastic slip is not specified, then a default value is computed for stick-stiffness calculations if necessary. The default value for the translational joint and the slot joint is set to $0.005 \cdot h$, where h is a characteristic length value computed from overall dimensions of the model. The value of h defaults to 1.0 if a characteristic length cannot be computed properly. The default value for the revolute joint is set to 0.001 radians.
- The frictional behavior is implemented using a penalty method. Thus, there will be relative elastic slip even when sticking conditions prevail. The amount of elastic slip depends on the value specified for elastic slip. In some cases, the default values may result in large elastic slip. Therefore, you should specify an amount of elastic slip that is appropriate for your model.

Stick-Stiffness

- A stick-stiffness value can be specified for controlling the behavior in the stick regime when friction behavior is specified. Use $TBOPT = SKx$, where x depends on the joint under consideration.
- If the stick-stiffness value is not specified, then the following procedure is adopted:
 - If both maximum force/moment and elastic slip are specified, then the stick-stiffness is calculated from these values.
 - If only maximum force/moment is specified, then a default elastic slip is computed and then the stick-stiffness is calculated.
 - If only the elastic slip is specified, then the stick-stiffness value is computed based on the current normal force/moment (Friction Coefficient * Normal Force or Moment/elastic-slip).

Interference Fit Force/Moment

- If the forces that are generated during a joint assembly have to be modeled, the interference fit force/moment can be specified using $TBOPT = Fx$, where x depends on the joint under consideration. This force/moment will contribute to the normal force/moment in friction calculations.

The appropriate $TBOPT$ labels (**TB** command) for each joint element type are shown in the table below:

$TBOPT$ Labels for Elements Supporting Coulomb Friction				
Friction Parameter	x-axis Revolute Joint	z-axis Revolute Joint	Slot Joint	Translational Joint
Static Friction	MUS4	MUS6	MUS1	MUS1
Exponential Friction Law	EXP4	EXP6	EXP1	EXP1
Max. Allowable Shear Force/Moment	TMX4	TMX6	TMX1	TMX1

Elastic Slip	SL4	SL6	SL1	SL1
Interference Fit Force/Moment	FI4	FI6	FI1	FI1
Stick-Stiffness	SK4	SK6	SK1	SK1

The following examples illustrate how to specify Coulomb friction parameters for various scenarios.

Example 1 Specifying a single value of coefficient of friction and other friction parameters for an x-axis revolute joint.

```
TB, JOIN, 1, , , MUS4      ! Label for friction coefficient
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , FI4       ! Label for interference fit force
TB, JOIN, 1, , , FI4       ! Label for interference fit force
TB, JOIN, 1, , , FI4       ! Label for interference fit force
TB, JOIN, 1, , , FI4       ! Label for interference fit force
TB, JOIN, 1, , , FI4       ! Label for interference fit force
TB, JOIN, 1, , , FI4       ! Label for interference fit force
TB, JOIN, 1, , , FI4       ! Label for interference fit force
TB, JOIN, 1, , , FI4       ! Label for interference fit force
```

Example 2 Specifying temperature dependent friction coefficient and other friction parameters for a z-axis revolution joint.

```
TB, JOIN, 1,2 , 1, MUS6    ! 2 temp points, 2 data points and label for friction coefficient
TBTEMP, 10                 ! 1st temperature
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
TB, JOIN, 1, , , SK4       ! Label for stick-stiffness
```

Example 3 Specifying the exponential law for friction and other friction parameters for a z-axis revolute joint.

```
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
TB, JOIN, 1, , , EXP6      ! Label for friction coefficient
```

Example 4 Specifying friction as a function of sliding velocity for a slot joint.

```
TB, JOIN, 1, , , 3, MUS1   ! Label for friction coefficient
TBPT, , 1.0, 0.15          ! Sliding velocity, coefficient of friction
TBPT, , 5.0, 0.10          ! Sliding velocity, coefficient of friction
TBPT, , 10.0, 0.09         ! Sliding velocity, coefficient of friction
TB, JOIN, 1, , , TMX1      ! Label for max allowable frictional force
TB, JOIN, 1, , , TMX1      ! Label for max allowable frictional force
TB, JOIN, 1, , , SL1       ! Label for elastic slip
TB, JOIN, 1, , , SL1       ! Label for elastic slip
TB, JOIN, 1, , , SL1       ! Label for elastic slip
TB, JOIN, 1, , , SL1       ! Label for elastic slip
TB, JOIN, 1, , , SL1       ! Label for elastic slip
TB, JOIN, 1, , , SL1       ! Label for elastic slip
```

4.16. Contact Friction

Contact friction (**TB,FRIC**) is a material property used with current-technology contact elements. It can be specified either through the coefficient of friction (MU) for [isotropic \(p. 205\)](#) or [orthotropic \(p. 205\)](#) friction models or as [user defined friction \(p. 207\)](#) properties.

For more information on how to specify friction in a contact analysis, see [Choosing a Friction Model in the *Contact Technology Guide*](#).

4.16.1. Isotropic Friction

Isotropic friction is applicable to 2-D and 3-D contact and is available for all contact elements. Use the **TB,FRIC** command with *TBOPT* = ISO to define isotropic friction, and specify the coefficient of friction MU on the **TBDATA** command. This is the recommended method for defining isotropic friction.

To define a coefficient of friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command. Suitable combinations of up to two fields can be used to define dependency, for example, temperature and sliding distance as shown below:

```
TB,FRIC,1,,ISO      ! Activate isotropic friction model
TBFIELD,TEMP,100.0  ! Define first value of temperature
TBFIELD,SLDI,0.1    ! Define first value of sliding distance
TBDATA,1,MU         ! Define coefficient of friction
TBFIELD,SLDI,0.5    ! Define second value of sliding distance
TBDATA,1,MU         ! Define coefficient of friction
TBFIELD,TEMP,200.0  ! Define second value of temperature
TBFIELD,SLDI,0.2    ! Define first value of sliding distance
TBDATA,1,MU         ! Define coefficient of friction
TBFIELD,SLDI,0.7    ! Define second value of sliding distance
TBDATA,1,MU         ! Define coefficient of friction
```

See [Understanding Field Variables \(p. 301\)](#) for more information on the interpolation scheme used for field-dependent material properties defined using **TBFIELD**.

To define a coefficient of friction that is dependent on temperature only, use the **TBTEMP** command as shown below:

```
TB,FRIC,1,2,,ISO    ! Activate isotropic friction model
TBTEMP,100.0         ! Define first temperature
TBDATA,1,MU          ! Define coefficient of friction at temp 100.0
TBTEMP,200.0         ! Define second temperature
TBDATA,1,MU          ! Define coefficient of friction at temp 200.0
```

Alternatively, you can use MU on the **MP** command to specify the isotropic friction. Use the **MPTEMP** command to define MU as a function of temperature.

Note that if the coefficient of friction is defined as a function of temperature, the program always uses the contact surface temperature as the primary variable (not the average temperature from the contact and target surfaces).

4.16.2. Orthotropic Friction

The orthotropic friction model uses two different coefficients of friction in two principal directions (see [Frictional Model in the *Theory Reference*](#) for details). It is applicable only to 3-D contact and is available for current-technology contact elements.

Issue the **TB,FRIC** command with *TBOPT* = ORTHO, EORTHO, or FORTHO to define orthotropic friction, and specify the coefficients of friction, MU1 and MU2, on the **TBDATA** command.

To define a coefficient of friction that is dependent on temperature, time, normal pressure, sliding distance, or sliding relative velocity, use the **TBFIELD** command. Suitable combinations of up to two

fields can be used to define dependency, for example, sliding relative velocity and normal pressure as shown below:

```
TB,FRIC,1,,ORTHO      ! Activate orthotropic friction model
TBFIELD,SLRV,10.0      ! Define first value of sliding relative velocity
TBFIELD,NPRE,200.0      ! Define first value of normal pressure
TBDATA,1,MU1,MU2       ! Define coefficients of friction
TBFIELD,NPRE,250.0      ! Define second value of normal pressure
TBDATA,1,MU1,MU2       ! Define coefficients of friction
TBFIELD,SLRV,20.0       ! Define second value of sliding relative velocity
TBFIELD,NPRE,150.0      ! Define first value of normal pressure
TBDATA,1,MU1,MU2       ! Define coefficients of friction
TBFIELD,NPRE,300.0      ! Define second value of normal pressure
TBDATA,1,MU1,MU2       ! Define coefficients of friction
```

See [Understanding Field Variables \(p. 301\)](#) for more information on the interpolation scheme used for field-dependent material properties defined using **TBFIELD**.

For *TBOPT* = ORTHO, the friction coordinate system is always attached to the contact element and rotates with the contact element.

For *TBOPT* = FORTHO, the friction coordinate system is essentially fixed in space and does not rotate as the contact element rotates.

The option *TBOPT* = EORTHO is similar to *TBOPT* = ORTHO, except when the frictional coefficients are defined as a function of sliding distance or sliding velocity. The difference occurs in the way the coefficients are interpolated. For *TBOPT* = ORTHO, the friction coefficient in each direction is a function of sliding distance or velocity in that direction only. For *TBOPT* = EORTHO, the friction coefficient in each direction depends upon the magnitude of total sliding or total velocity, thus causing sliding in one direction to affect the friction coefficients in both directions.

To define a coefficient of friction that is dependent on temperature only, use the **TBTEMP** command as shown below:

```
TB,FRIC,1,2,,ORTHO    ! Activate orthotropic friction model
TBTEMP,100.0           ! Define first temperature
TBDATA,1,MU1,MU2       ! Define coefficients of friction at temp 100.0
TBTEMP,200.0           ! Define second temperature
TBDATA,1,MU1,MU2       ! Define coefficients of friction at temp 200.0
```

Note that if the coefficient of friction is defined as a function of temperature, the program always uses the contact surface temperature as the primary variable (not the average temperature from the contact and target surfaces).

4.16.3. Redefining Friction Between Load Steps

If the friction behavior changes between initial loading and secondary loading (for example, during cyclic loading of seabed pipelines), you can reissue the **TB,FRIC** command between load steps to define new values for the coefficient of friction. This is true for both temperature-dependent friction (isotropic or orthotropic) defined via the **TBTEMP** command and field-dependent friction (isotropic or orthotropic) defined via the **TBFIELD** command. The following example shows the latter case:

```
TB,FRIC,1,,ORTHO      !Activate orthotropic friction model
TBFIELD,SLDI,0.        !Define initial curve for coefficient of friction
TBDATA,1,0.0,0.0
TBFIELD,SLDI,0.25
TBDATA,1,0.0,1.25
TBFIELD,SLDI,0.5
TBDATA,1,0.0,1.0
TBFIELD,SLDI,20.
```



```

TBDATA,1,0.0,1.1

/SOLUTION
!* LOAD STEP 1
...
TIME,1
SOLVE

TB,FRIC,1,,,ORTHO      !Activate orthotropic friction model
TBFIELD,SLDI,0.         !Define secondary curve for coefficient of friction
TBDATA,1,0.0,20.0
TBFIELD,SLDI,1.1
TBFIELD,SLDI,20.25
TBDATA,1,0.0,0.0
TBFIELD,SLDI,20.5
TBDATA,1,0.0,0.8
TBFIELD,SLDI,21
TBDATA,1,0.0,0.7
TBFIELD,SLDI,35
TBDATA,1,0.0,0.75

!* LOAD STEP 2
...
TIME,2
SOLVE

```

4.16.4. User-Defined Friction

As an alternative to the program-supplied friction models, you can define your own friction model with the user programmable friction subroutine, USERFRIC. The frictional stresses can be defined as a function of variables such as slip increments, sliding rate, temperature, and other arguments passed into the subroutine. You can specify a number of properties or constants associated with your friction model, and you can introduce extra solution-dependent state variables that can be updated and used within the subroutine. User-defined friction is applicable to 2-D and 3-D contact elements.

To specify user-defined friction, use the **TB,FRIC** command with *TBOPT* = USER and specify the friction properties on the **TBDATA** command, as shown below. Also, use the USERFRIC subroutine to program the friction model.

```

TB,FRIC,1,,2,USER      ! Activate user defined friction model; NPTS = 2
TBDATA,1,PROP1,PROP2   ! Define friction properties

```

Field variables specified with the **TBFIELD** command are not available for **TB,FRIC,,,,USER**.

For detailed information on using the USERFRIC subroutine, see [Writing Your Own Friction Law \(USERFRIC\)](#) in the *Contact Technology Guide*.

4.17. Contact Interaction

Contact interaction (**TB,INTER**) can be used to specify the type of interaction between general contact surfaces. These interactions are defined via the **GCDEF** command. **TB,INTER** can also be used to specify a [user-defined interaction](#) (p. 209) for pair-based contact elements or for general contact definitions.

4.17.1. Interaction Options for General Contact Definitions

Contact interactions for general contact definitions are specified via the **TB,INTER** command instead of using KEYOPT(12) (as for pair-based contact definitions). The interaction option is specified by the *TBOPT* field as shown in the table below.

TBOPT Label	Interaction Behavior	Description
STANDARD	Standard unilateral contact	Normal pressure equals zero if separation occurs. (Similar to KEYOPT(12) = 0 for pair-based contact elements.)
ROUGH	Rough (no sliding)	Perfectly rough frictional contact where there is no sliding. (Similar to KEYOPT(12) = 1 for pair-based contact elements.)
NOSEPE	No separation (sliding permitted)	No separation contact in which the target and contact surfaces are tied for the remainder of the analysis once contact is established (although sliding is permitted). (Similar to KEYOPT(12) = 2 for pair-based contact elements.)
BONDED	Bonded contact (no separation, no sliding)	Bonded contact in which the target and contact surfaces are bonded in all directions (once contact is established) for the remainder of the analysis. (Similar to KEYOPT(12) = 3 for pair-based contact elements.)
ANOSEP	No separation (always)	No separation contact in which contact detection points that are either initially inside the pinball region or that once involve contact always attach to the target surface along the normal direction to the contact surface (sliding is permitted). (Similar to KEYOPT(12) = 4 for pair-based contact elements.)
ABOND	Bonded (always)	Bonded contact in which contact detection points that are either initially inside the pinball region or that once involve contact always attach to the target surface along the normal and tangent directions to the contact surface (fully bonded). (Similar to KEYOPT(12) = 5 for pair-based contact elements.)
IBOND	Bonded (initial contact)	Bonded contact in which the contact detection points that are initially in a closed state will remain attached to the target surface, and the contact detection points that are initially in an open state will remain open throughout the analysis. (Similar to KEYOPT(12) = 6 for pair-based contact elements.)

All of the above options use one material constant (C1) on the **TBDATA** command. The value of C1 defines the effect of initial penetration or gap, as described in the table below.

C1 Value	Description
0 [1 (p. 209)]	Include both initial geometrical penetration or gap and offset. (Similar to KEYOPT(9) = 0 for pair-based contact elements.)
1 [2 (p. 209)]	Exclude both initial geometrical penetration or gap and offset. (Similar to KEYOPT(9) = 1 for pair-based contact elements.)
2	Include both initial geometrical penetration or gap and offset, but with ramped effects. (Similar to KEYOPT(9) = 2 for pair-based contact elements.)
3	Include offset only (exclude initial geometrical penetration or gap). (Similar to KEYOPT(9) = 3 for pair-based contact elements.)
4	Include offset only (exclude initial geometrical penetration or gap), but with ramped effects. (Similar to KEYOPT(9) = 4 for pair-based contact elements.)
5	Include offset only (exclude initial geometrical penetration or gap) regardless of the initial contact status (near-field or closed). (Similar to KEYOPT(9) = 5 for pair-based contact elements.)

C1 Value	Description
6	Include offset only (exclude initial geometrical penetration or gap), but with ramped effects regardless of the initial contact status (near-field or closed). (Similar to KEYOPT(9) = 6 for pair-based contact elements.)

1. C1 = 0 is the default for all general contact interaction types if **TBDATA** is issued but C1 is not specified.
2. C1 = 1 is the default for all general contact interaction types if **TBDATA** is not issued.

Note that the effects of C1 input are dependent on which *TBOPT* label was previously defined. The indicated initial gap effect is considered only if *TBOPT* = ANOSE or *TBOPT* = ABOND is defined.

4.17.2. User-Defined Interaction

As an alternative to the program-supplied interface behaviors, you can define your own interaction model with the user programmable subroutine, USERINTER. You can specify a number of properties or constants associated with your interaction model, and you can introduce extra solution-dependent state variables that can be updated and used within the subroutine. User-defined interaction is applicable to 2-D and 3-D contact elements used in pair-based contact definitions or general contact definitions.

After writing a USERINTER subroutine to program your interaction model, you incorporate the model in your analysis by using the command **TB,INTER** with *TBOPT* = USER and specifying the interaction properties on the **TBDATA** command as shown below.

```
TB,INTER,1,,2,USER      ! Activate user defined contact interaction; NPTS = 2
TBDATA,1,PROP1,PROP2    ! Define interaction properties
```

For detailed information on using the USERINTER subroutine, see [Defining Your Own Contact Interaction \(USERINTER\)](#) in the *Contact Technology Guide*.

4.18. Cohesive Material Law

Cohesive zone materials can be used with interface elements (INTER nnn) and contact elements (CON-TAnnn), as described here:

- 4.18.1. Exponential Cohesive Zone Material for Interface Elements and Contact Elements
- 4.18.2. Bilinear Cohesive Zone Material for Interface Elements and Contact Elements
- 4.18.3. Viscous Regularization of Cohesive Zone Material for Interface Elements and Contact Elements
- 4.18.4. Cohesive Zone Material for Contact Elements
- 4.18.5. Post-Debonding Behavior at the Contact Interface

Also see [User-Defined Cohesive Material \(UserCZM\)](#) (p. 216) and [Material Model Element Support](#) (p. 5) for this material model.

For more information about cohesive zone materials, see [Cohesive Zone Material \(CZM\) Model in the Theory Reference](#).

4.18.1. Exponential Cohesive Zone Material for Interface Elements and Contact Elements

Interface elements and contact elements allow exponential cohesive zone materials to be used for simulating interface delamination and other fracture phenomena. To define exponential material behavior, issue the **TB,CZM,,,EXPO** command, then specify the following material constants via the **TBDATA** command:

Constant	Meaning	Property
C1	$\sigma_{\max}$	Maximum normal traction at the interface
C2	δ_n	Normal separation across the interface where the maximum normal traction is attained
C3	δ_t	Shear separation where the maximum shear traction is attained

To define a temperature dependent material, use the **TBTEMP** command as shown below:

```
TB,CZM,1,2,,EXPO      ! Activate exponential material model
TBTEMP,100.0          ! Define first temperature
TBDATA,1, max, n, t    ! Define material constants at temp 100.0
TBTEMP,200.0          ! Define second temperature
TBDATA,1, max, n, t    ! Define material constants at temp 200.0
```

4.18.2. Bilinear Cohesive Zone Material for Interface Elements and Contact Elements

Interface elements and contact elements allow bilinear cohesive zone materials to be used for simulating interface delamination and other fracture phenomena. To define bilinear material behavior, issue the **TB,CZM,,,BILI** command, then specify the following material constants via the **TBDATA** command:

Constant	Meaning	Property
C1	$\sigma_{\max}$	Maximum normal traction
C2	δ_n^c	Normal displacement jump at the completion of debonding
C3	$\tau_{\max}$	Maximum tangential traction
C4	δ_t^c	Tangential displacement jump at the completion of debonding
C5	α	Ratio of δ_n^* to δ_n^c , or ratio of δ_t^* to δ_t^c
C6 [1]	β	Non-dimensional weighting parameter

1. C6 must be the constant at all temperatures.

To define a temperature-dependent material, issue the **TBTEMP** command as shown in the following example input fragment:

```
TB,CZM,1,2,,BILI      ! Activate bilinear CZM material model
TBTEMP,100.0          ! Define first temperature
!   Define Mode I dominated material constants at temp 100.0:
!TBDATA,1, max,  $\delta_n^c$ , - max,  $\delta_t^c$ ,
!
```

```

TBTEMP,200.0      ! Define second temperature
!   Define Mode I dominated material constants at temp 200.0:
TBDATA,1, max,  $\delta_n^c$ , - max,  $\delta_t^c$ ,

```

Debonding Interface Modes

Three modes of interface debonding make up bilinear CZM law:

Case	Input on the TBDATA command as follows:
Mode I Dominated	$C1, C2, C3, C4, C5$ (where $C3 = -\tau_{\max}$)
Mode II Dominated	$C1, C2, C3, C4, C5$ (where $C1 = -\sigma_{\max}$)
Mixed-Mode	$C1, C2, C3, C4, C5, C6$ (where $C1 = \sigma_{\max}$ and $C3 = \tau_{\max}$)

4.18.3. Viscous Regularization of Cohesive Zone Material for Interface Elements and Contact Elements

Interface elements and contact elements allow viscous regularization to be used for stabilizing interface delamination. Viscous regularization is valid with the [exponential cohesive zone material model](#) (**TBOPT** = EXPO) (p. 210) and the [bilinear cohesive zone material model](#) (**TBOPT** = BILI) (p. 210).

To define viscous regularization parameters, issue the **TB,CZM,,,VREG** command, then specify the following material constant via the **TBDATA** command:

Constant	Meaning	Property
C1	ζ	Damping coefficient

To define a temperature-dependent material, use the **TBTEMP** command as shown in the following example input fragment:

```

TBTEMP,100.0      ! Define first temperature
TBDATA,1,C1        ! Define damping coefficient at temp 100.0
TBTEMP,200.0      ! Define second temperature
TBDATA,1,C1        ! Define damping coefficient at temp 200.0

```

For more information, see [Viscous Regularization](#) in the [Mechanical APDL Theory Reference](#).

4.18.4. Cohesive Zone Material for Contact Elements

To model interface delamination, also known as [debonding](#), the contact elements support an additional cohesive zone material model with bilinear behavior. This model allows two ways to specify material data.

Bilinear Material Behavior with Traction and Separation Distances

To define bilinear material behavior with tractions and separation distances, issue the **TB,CZM,,,CBDD** command, then specify the following material constants via the **TBDATA** command:

Constant	Meaning	Property
C1	$\sigma_{\max}$	Maximum normal contact stress [1]
C2	δ_n^c	Contact gap at the completion of debonding

Constant	Meaning	Property
C3	$\tau_{\max}$	Maximum equivalent tangential contact stress [1]
C4	δ_t^c	Tangential slip at the completion of debonding
C5	η	Artificial damping coefficient
C6	β	Flag for tangential slip under compressive normal contact stress; must be 0 (off) or 1 (on)
C7	α	Power law exponent for mixed-mode debonding (defaults to 2)

1. For contact elements using the force-based model (see the description of KEYOPT(3) for [CONTA175](#) and [CONTA177](#)), input a contact force value for this quantity.

To define a temperature dependent material, use the **TBTEMP** command as shown below:

```
TB,CZM,1,2,,CBDD      ! Activate bilinear material model with tractions
                        ! and separation distances
TBTEMP,100.0          ! Define first temperature

TBDATA,1, max,  $\delta_n^c$ , max,  $\delta_t^c$ , , ,      ! Define material constants at temp 100.0
TBDATA,7,
TBTEMP,200.0          ! Define second temperature

TBDATA,1, max,  $\delta_n^c$ , max,  $\delta_t^c$ , , ,      ! Define material constants at temp 200.0
TBDATA,7,
```

Bilinear Material Behavior with Tractions and Critical Fracture Energies

Use the **TB,CZM** command with **TBOPT** = CBDE to define bilinear material behavior with tractions and critical fracture energies, and specify the following material constants using the **TBDATA** command.

Constant	Meaning	Property
C1	$\sigma_{\max}$	Maximum normal contact stress [1]
C2	G_{cn}	Critical fracture energy density (energy/area) for normal separation [2]
C3	$\tau_{\max}$	Maximum equivalent tangential contact stress [1]
C4	G_{ct}	Critical fracture energy density (energy/area) for tangential slip [2]
C5	η	Artificial damping coefficient
C6	β	Flag for tangential slip under compressive normal contact stress; must be 0 (off) or 1 (on)
C7	α	Power law exponent for mixed-mode debonding (defaults to 2)

1. For contact elements using the force-based model (see the description of KEYOPT(3) for [CONTA175](#) and [CONTA177](#)), input a contact force value for this quantity.
2. For contact elements using the force-based model (see the description of KEYOPT(3) for [CONTA175](#) and [CONTA177](#)), this quantity is critical fracture energy.

To define a temperature dependent material, use the **TBTEMP** command as shown in the following example input fragment:

```

TB,CZM,1,2,,CBDE           ! Activate bilinear material model with
                             ! tractions and fracture energies
TBTEMP,100.0               ! Define first temperature
TBDATA,1,max,Gcn,max,Gct,    ! Define material constants at temp 100.0
TBDATA,7,
TBTEMP,200.0               ! Define second temperature
TBDATA,1,max,Gcn,max,Gct,    ! Define material constants at temp 200.0
TBDATA,7,

```

Debonding Modes

Debonding involves separation of surfaces forming an interface. The direction of separation determines the debonding mode. The program detects the debonding mode based on material data that you input for normal and tangential directions:

- **Mode I debonding** involves separation normal to the interface. It is activated by inputting data items C1, C2, and C5 on the **TBDATA** command.
- **Mode II debonding** involves slip tangent to the interface. It is activated by inputting data items C3, C4, and C5 on the **TBDATA** command.
- **Mixed-mode debonding** involves both normal separation and tangential slip. It is activated by inputting data items C1, C2, C3, C4, C5, C6, and C7 on the **TBDATA** command.

Debonding

When friction is defined between contact surfaces undergoing debonding, tangential stress is calculated as the maximum between the tangential stress as governed by the debonding model and the tangential stress as governed by the friction law.

4.18.5. Post-Debonding Behavior at the Contact Interface

When the cohesive zone material defined at a contact interface is completely debonded, the contact behavior at that interface is changed to standard contact (KEYOPT(12) = 0) by default. This default behavior can be changed for certain CZM materials.

For the cohesive zone materials with bilinear material behavior (*TBOPT* = CBDD, CBDE or BILL on the **TB** command), you can specify that the cohesive zone interface be “healed” if the surfaces come into contact again after debonding. To activate this option, use the **TBFIELD,CYCLE** command to define the CZM material as a function of healing cycle number. You can use multiple **TBFIELD** commands to specify the material properties for any number of healing cycles, but be sure to start with a cycle number of zero.

For example, the following commands specify healing of the CZM interface if the contact surfaces come into contact after they are completely debonded:

```

TB,CZM,1,,,CBDE           ! Activate the CBDE bilinear material behavior
TBFIELD,CYCLE,0           ! Initial CZM definition (before healing)
TBDATA,1,max,Gcn,max,Gct,    ! CZM properties
TBFIELD,CYCLE,1           ! CZM definition for first healing cycle
TBDATA,1,...              ! CZM properties to be used after first healing

```


When the contact interface is completely debonded and the surfaces come into contact again, the [debonding parameter](#) is set to 0 thus effectively healing the CZM. The healing cycle is incremented by one and the appropriate material data is interpolated for this healing cycle.

This healing option is only available when one of the supported cohesive zone materials is used with contact elements. It is not available when a cohesive zone material is used with interface elements.

4.19. Contact Surface Wear

Contact surface wear can be simulated by defining a wear model (**TB,WEAR**) as a material assigned to contact elements. Two options are available: the Archard wear model and a user-defined wear model (the **USERWEAR** subroutine). These two options are discussed below. For additional information, see [Contact Surface Wear](#) in the *Contact Technology Guide*.

4.19.1. Archard Wear Model

The Archard wear model defines the rate of wear as a function of contact pressure, sliding velocity, and material hardness. If a wear direction is not specified, the default direction of wear is opposite to the contact normal.

The Archard model is defined by the **TB,WEAR** command with **TBOPT** = ARCD. The material constants required by the model are specified as data items C1 through C4 on the **TBDATA** command. You can also specify the wear direction (C6 to C8) if desired.

Constant	Meaning
C1	Wear coefficient, K
C2	Material hardness, H
C3	Pressure exponent, m
C4	Velocity exponent, n
C5	Optional flag to control how the wear increment is calculated (see below)
C6	Direction cosine nx (with respect to the global X axis) for the wear direction
C7	Direction cosine ny (with respect to the global Y axis) for the wear direction
C8	Direction cosine nz (with respect to the global Z axis) for the wear direction

The fifth constant, C5, can be used to specify additional controls:

- Set C5 to 1 to base the wear calculation on nodal stresses. The nodal stresses of the solid element underlying the contact element are used to calculate traction along the contact normal direction. The traction value is used instead of contact pressure to calculate the amount of wear.
- Set C5 to 10 or 11 to average the wear increment over the contact area of the contact pair. Use C5 = 10 to base the wear calculation on contact pressure; use C5 = 11 to base the wear calculation on nodal stress.
- Set C5 to -99 to calculate wear for postprocessing purposes only. The Archard model is used to calculate the wear, but the contact nodes are not moved; thus wear is just a postprocessing variable and does not affect the solution.

Use the **TBFIELD** command to define the constants as a function of temperature and/or time. A sample input with constants as a function of time is shown below:


```

TB,WEAR,1,,,ARCD      ! Activate Archard wear model
TBFIELD,TIME,0        ! Define the first value of time
TBDATA,1,K,H,m,n      ! Define wear material constants for the first value of time
TBFIELD,TIME,1        ! Define the second value of time
TBDATA,1,K,H,m,n      ! Define wear material constants for the second value of time

```

Alternatively, you can use the **TBTEMP** command to define the constants as a function of temperature alone. A sample input is shown below:

```

TB,WEAR,1,,,ARCD      ! Activate Archard wear model
TBTEMP,100            ! Define the first value of temperature
TBDATA,1,K,H,m,n      ! Define wear material constants for the first value of temperature
TBTEMP,200            ! Define the second value of temperature
TBDATA,1,K,H,m,n      ! Define wear material constants for the second value of temperature

```

4.19.2. User-Defined Wear Model

As an alternative to the Archard wear model, you can define your own wear model via the user-programmable subroutine, **USERWEAR**. This subroutine allows you to define the increment of wear for a substep. The default wear direction is opposite to the contact normal. However, you can redefine it inside of **USERWEAR**.

The user-defined wear model is activated by the **TB,WEAR** command with **TBOPT = USER**. A sample command input is shown below:

```

TB,WEAR,1,,,4,USER      ! Activate user defined wear model that requires 4 (NPTS = 4) constants
TBDATA,1,C1,C2,C3,C4    ! Define the wear model constants

```

4.20. Custom Material Models

The following topics about creating your own custom material models are available:

[4.20.1. Creating a User-Defined Material Model](#)

[4.20.2. User-Defined Cohesive Material \(UserCZM\)](#)

[4.20.3. Using State Variables with User-Defined Materials](#)

4.20.1. Creating a User-Defined Material Model

Two routines are available for creating your own material models:

[4.20.1.1. User-Defined Material Model \(UserMat\)](#)

[4.20.1.2. User-Defined Thermal Material Model \(UserMatTh\)](#)

4.20.1.1. User-Defined Material Model (UserMat)

The user-defined material option (**TB,USER**) describes input parameters for defining your own material model via the **UserMat** subroutine, which defines any material type except incompressible materials. For more information, see [Subroutine UserMat \(Creating Your Own Material Model\)](#).

Input for the user-defined option is determined by constants which you define. **TB,USER** initializes the constant table. The constants are defined via **TBDATA** (six per command). The number of constants can be any combination of the number of temperatures (**NTEMP**) and the number of data points per temperature (**NPTS**), to a maximum of $NTEMP \times NPTS = 1000$.

Define temperatures via **TBTEMP**.

Example 4.37: Input for a User-Defined Material

```

TB,USER,1,2,4          ! Set material 1 as a user-defined
                        ! material with two temperatures
                        ! and four data points at each
                        ! temperature point.
TBTEMP,1.0             ! First temperature.
TBDATA,1,19e5,0.3,1e3,100, ! Four material constants for
                        ! first temperature.
TBTEMP,2.0             ! Second temperature.
TBDATA,1,21e5,0.3,2e3,100, ! Four material constants for
                        ! second temperature.

```

For information about state variable support, see [Using State Variables with the UserMat Subroutine](#) (p. 217).

4.20.1.2. User-Defined Thermal Material Model (UserMatTh)

The user-defined thermal material option (**TB,USER**) describes input parameters for defining your own thermal material model via the UserMatTh subroutine. For more information, see [Subroutine UserMatTh \(Creating Your Own Thermal Material Model\)](#).

Input for the user-defined option is determined by constants which you define. **TB,USER** initializes the constant table. The constants are defined via **TBDATA** (six per command). The number of constants can be any combination of the number of temperatures (*NTEMP*) and the number of data points per temperature (*NPTS*), to a maximum of $NTEMP \times NPTS = 1000$.

Define temperatures via **TBTEMP**.

Example 4.38: Input for a User-Defined Thermal Material

```

TB,USER,1,2,4          ! Set material 1 as a user-defined
                        ! material with two temperatures
                        ! and four data points at each
                        ! temperature point.
TBTEMP,1.0             ! First temperature.
TBDATA,1,19e5,0.3,1e3,100, ! Four material constants for
                        ! first temperature.
TBTEMP,2.0             ! Second temperature.
TBDATA,1,21e5,0.3,2e3,100, ! Four material constants for
                        ! second temperature.

```

In a coupled-field analysis with structural and thermal degrees of freedom, use **TB,USER** with **TBOPT = THERM** to specify user-defined thermal material properties independently of a user-defined structural material model. The THERM option is only available with elements **SOLID226** and **SOLID227**.

For information about state variable support, see [Using State Variables with the UserMatTh Subroutine](#) (p. 217).

4.20.2. User-Defined Cohesive Material (UserCZM)

Support is available for creating a user-defined cohesive material (**TB,CZM,,,USER**) via the UserCZM subroutine. The subroutine supports interface elements (**INTERnnn**) only. For more information, see [Subroutine userCZM \(Defining Your Own Cohesive Zone Material\)](#) in the *Programmer's Reference*.

Input is determined by user-specified constants (**TBDATA**). Up to six constants can be defined per **TBDATA** command. The number of constants can be any combination of the number of temperatures (*NTEMP*) and the number of data points per temperature (*NPTS*), such that $NTEMP \times NPTS \leq 1000$.

Example 4.39: Input for a User-Defined Cohesive Material Law

```
TB,CZM,1,2,4,USER      ! Set material 1 as a user-defined
                        ! cohesive material option with two temperatures
                        ! and four data points at each
                        ! temperature point.
TBTEMP,1.0              ! First temperature.
TBDATA,1,19e5,0.3,1e3,100, ! Four cohesive material constants for
                        ! first temperature.
TBTEMP,2.0              ! Second temperature.
TBDATA,1,21e5,0.3,2e3,100, ! Four cohesive material constants for
                        ! second temperature.
```

Contact elements do not support a user-defined cohesive material. However, a similar capability can be achieved by defining your own contact interface behavior via the `userinter` subroutine. For more information, see [Defining Your Own Contact Interaction \(USERINTER\)](#) in the *Contact Technology Guide*.

For information about state variable support, see [Using State Variables with the UserCZM Subroutine](#) (p. 218).

4.20.3. Using State Variables with User-Defined Materials

The following topics related to using state variables with subroutines for creating your own custom materials are available:

[4.20.3.1. Using State Variables with the UserMat Subroutine](#)

[4.20.3.2. Using State Variables with the UserMatTh Subroutine](#)

[4.20.3.3. Using State Variables with the UserCZM Subroutine](#)

4.20.3.1. Using State Variables with the UserMat Subroutine

The `UserMat` subroutine for creating [user-defined material models](#) (p. 215) supports state variables. To use them, initialize the constant table (**TB,STATE**), then define the constants (**TBDATA**).

Example 4.40: Initializing the Values of State Variables for a User-Defined Material

```
TB,STATE,1,,4,          ! Define material 1, which
                        ! has four state variables.
TBDATA,1,C1,C2,C3,C4,   ! Initialize the four state variables.
```

You can define a maximum of 1000 state variables ($NPTS = 1000$).

4.20.3.2. Using State Variables with the UserMatTh Subroutine

The `UserMatTh` subroutine for creating [user-defined thermal material models](#) (p. 216) supports state variables. To use them, initialize the constant table (**TB,STATE**), then define the constants (**TBDATA**).

Example 4.41: Initializing the Values of State Variables for a User-Defined Material

```
TB,STATE,1,,4,      ! Define material 1, which
                    ! has four state variables.
TBDATA,1,C1,C2,C3,C4, ! Initialize the four state variables.
```

You can define a maximum of 1000 state variables ($NPTS = 1000$).

4.20.3.3. Using State Variables with the UserCZM Subroutine

The [UserCZM subroutine](#) for creating [user-defined CZM models](#) (p. 216) supports state variables. To use them, initialize the constant table (**TB,STATE**), then define the initial constants (**TBDATA**).

By default, the initial values of state variables are set to zero.

Example 4.42: Initializing the Values of State Variables for a User-Defined Cohesive Material Law

```
TB,STATE,1,,4,      ! Define material 1, which
                    ! has four state variables.
TBDATA,1,C1,C2,C3,C4, ! Initialize the four state variables.
```

4.21. Material Strength Limits

Material strength limits represent maximum stresses or strains that a material can sustain. This data table defines the strength limits and other related constants required for computing failure criteria (FC) index of a material under various loading conditions. Specify a *TBOPT* value on the **TB,FCLI** command to correspond to the stress limits ($TBOPT = 1$) or strain limits ($TBOPT = 2$). The following table lists the coefficient values that are addressed for the available *TBOPT* values:

Strength Limit Constants	<i>TBOPT</i> = 1	<i>TBOPT</i> = 2
	Stress Limits ($NPTS = 16$)	Strain Limits ($NPTS = 9$)
C1	XTEN -- Allowable tensile stress in material X-direction (must be positive)	XTEN -- Allowable tensile strain in material X-direction (must be positive)
C2	XCMP -- Allowable compressive stress in material X-direction (default to the negative of XTEN)	XCMP -- Allowable compressive strain in material X-direction (default to the negative of XTEN)
C3	YTEN -- Allowable tensile stress in material Y-direction (must be positive)	YTEN -- Allowable tensile strain in material Y-direction (must be positive)
C4	YCMP -- Allowable compressive stress in material Y-direction (default to the negative of YTEN)	YCMP -- Allowable compressive strain in material Y-direction (default to the negative of YTEN)
C5	ZTEN -- Allowable tensile stress in material Z-direction (must be positive)	ZTEN -- Allowable tensile strain in material Z-direction (must be positive)
C6	ZCMP -- Allowable compressive stress in material Z-direction (default to the negative of ZTEN)	ZCMP -- Allowable compressive strain in material Z-direction (default to the negative of ZTEN)

Strength Limit Constants	<i>TBOPT</i> = 1	<i>TBOPT</i> = 2
	Stress Limits (NPTS = 16)	Strain Limits (NPTS = 9)
C7	XY -- Allowable XY shear stress (must be positive)	XY -- Allowable XY shear strain (must be positive)
C8	YZ -- Allowable YZ shear stress (must be positive)	YZ -- Allowable YZ shear strain (must be positive)
C9	XZ -- Allowable XZ shear stress (must be positive)	XZ -- Allowable XZ shear strain (must be positive)
C10	XYCP -- XY coupling coefficient for Tsai-Wu strength index (default = -1.0)	--
C11	YZCP -- YZ coupling coefficient for Tsai-Wu failure index (default = -1.0)	--
C12	XZCP -- XZ coupling coefficient for Tsai-Wu failure index (default = -1.0)	--
C13	XZIT -- XZ tensile inclination parameter for Puck failure index (default = 0.0)	--
C14	XZIC -- XZ compressive inclination parameter for Puck failure index (default = 0.0)	--
C15	YZIT -- YZ tensile inclination parameter for Puck failure index (default = 0.0)	--
C16	YZIC -- YZ compressive inclination parameter for Puck failure index (default = 0.0)	--
C17	G1G2 -- Fracture toughness ratio between GI (mode I) and GII (mode II)	--
C18	ETAL -- Longitudinal friction coefficient	--
C19	ETAT -- Transverse friction coefficient	--
C20	ALP0 -- Fracture angle under pure transverse compression	--

To determine physical failure criteria in unidirectional fiber-reinforced composite materials, including Puck and Hashin, LaRc03, and LaRc04 criteria, always define the reinforced fiber direction as the material X direction.

The following table summarizes the applicable strength-limit constants for each failure criterion:

Strength Limit Constants	Max Strain Criterion	Max Stress Criterion	Tsai-Wu Strength Ratio	Puck Criterion	Hashin Criterion	LaRc03/04 Criterion	User-Defined
C1	Y	Y	Y	Y	Y	Y	Y
C2	Y	Y	Y	Y	Y	Y	Y
C3	Y	Y	Y	Y	Y	Y	Y
C4	Y	Y	Y	Y	Y	Y	Y
C5	Y	Y	Y	--	--	--	Y
C6	Y	Y	Y	--	--	--	Y
C7	Y	Y	Y	Y	Y	Y	Y
C8	Y	Y	Y	--	Y	--	Y
C9	Y	Y	Y	--	--	--	Y
C10	--	--	Y	--	--	--	Y
C11	--	--	Y	--	--	--	Y
C12	--	--	Y	--	--	--	Y
C13	--	--	--	Y	--	--	Y
C14	--	--	--	Y	--	--	Y
C15	--	--	--	Y	--	--	Y
C16	--	--	--	Y	--	--	Y
C17	--	--	--	--	--	Y	Y
C18	--	--	--	--	--	Y	Y
C19	--	--	--	--	--	Y	Y
C20	--	--	--	--	--	Y	Y

4.22. Material Damage

The following topics concerning material damage initiation and evolution are available:

[4.22.1. Damage Initiation Criteria](#)

[4.22.2. Damage Evolution Law](#)

4.22.1. Damage Initiation Criteria

This data table defines the criteria type for determining the onset of material damage under loading.

Specify the *TBOPT* value on the **TB,DMGI** command to correspond to failure criteria (*TBOPT* = 1 or FCRT).

The following table shows the coefficient values addressed for the available *TBOPT* values:

Constants	<i>TBOPT</i> = 1 or FCRT
	<i>NPTS</i> = 4
C1	Failure criteria type [1] for the tensile fiber failure mode

Constants	<i>TBOPT = 1 or FCRT</i>
	<i>NPTS = 4</i>
C2	Failure criteria type [1] for the compressive fiber failure mode
C3	Failure criteria type [1] for the tensile matrix failure mode
C4	Failure criteria type [1] for the compressive matrix failure mode

1. The available failure criteria are as follows:

- 1 -- Maximum strain
- 2 -- Maximum stress
- 3 -- Puck
- 4 -- Hashin
- 5 -- LaRc03
- 6 -- LaRc04
- 11 -- User-defined #1
- ...
- 19 -- User-defined #9

To complete the material damage definition, it is also necessary to specify a compatible material damage evolution law (**TB,DMGE**). Without a damage evolution law, the damage-initiation criteria have no effect on the material. The following table summarizes the compatible damage evolution laws with specific damage-initiation criteria:

TB,DGMI Input		Compatible TB,DMGE Option
TBOPT value	C1, C2, C3, C4	
1	1, 2, ..., 19	1 or MPDG
1	4 (Hashin failure criteria only)	2 or CDM

4.22.2. Damage Evolution Law

This data table defines the material damage evolution law (or the way a material degrades) following the initiation of damage (**TB,DMGE**).

Specify the **TBOPT** value on the **TB,DMGE** command to correspond to the instant stiffness reduction (**TBOPT = 1 or MPDG**).

The following table shows the coefficient values addressed for the available **TBOPT** values:

Constants	<i>TBOPT = 1 or MPDG</i>
	<i>NPTS = 4</i>
C1	Tensile fiber stiffness reduction [1]

Constants	TBOPT = 1 or MPDG
	NPTS = 4
C2	Compressive fiber stiffness reduction [1]
C3	Tensile matrix stiffness reduction [1]
C4	Compressive matrix stiffness reduction [1]

1. The allowable values of C1, C2, C3, and C4 are between 0 and 1, where 0 = no reduction in material stiffness in the affected mode after damage initiation, and 1 = complete stiffness loss in the affected mode.

Constants	TBOPT = 2 or CDM
	NPTS = 8
C1	Energy dissipated per unit area from tensile fiber damage [1]
C2	Viscous damping coefficient for tensile fiber damage [2]
C3	Energy dissipated per unit area from compressive fiber damage [1]
C4	Viscous damping coefficient for compressive fiber damage [2]
C5	Energy dissipated per unit area from tensile matrix damage [1]
C6	Viscous damping coefficient for tensile matrix damage [2]
C7	Energy dissipated per unit area from compressive matrix damage [1]
C8	Viscous damping coefficient for compressive matrix damage [2]

1. Energies dissipated per unit area G_c are specified individually for all damage modes (fiber tension, fiber compression, matrix tension, and matrix compression). For a specific damage mode, G_c is given by:

$$G_c = \int_0^{u_e^f} \sigma_e du_e$$

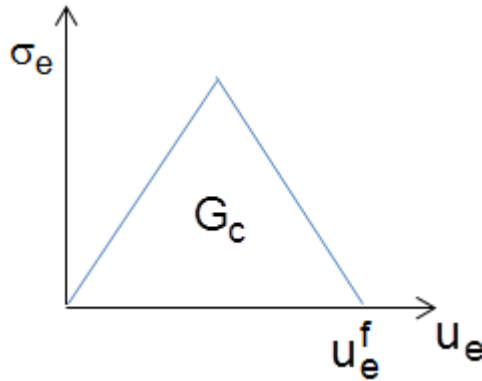
where:

σ_e = equivalent stress. For simple uniaxial stress state, equivalent stress is the actual stress.

u_e = equivalent displacement. For simple uniaxial stress state, $u_e = \epsilon \times L_c$
(where, L_c is the length of the element in the stress direction)

u_e^f = ultimate equivalent displacement, where total material stiffness is lost for the specific mode

For complex stress state, the equivalent stresses and strains are calculated based on Hashin failure criteria.



2. Viscous damping coefficients η are also specified respectively for all four damage modes. For a specific damage mode, the damage evolution is regularized as follows:

$$\dot{d}_{t+\Delta t} = \frac{\eta}{\eta + \Delta t} \dot{d}_t + \frac{\Delta t}{\eta + \Delta t} \dot{d}_{t+\Delta t}$$

where:

$\dot{d}_{t+\Delta t}$ = regularized damage variable at current time. $d_{t+\Delta t}$ is used for material degradation

$\dot{d}_t$ = regularized damage variable at the end of the last substep

$d_{t+\Delta t}$ = unregularized current damage variable

To complete the material damage definition, it is also necessary to specify a compatible material damage initiation criterion (**TB,DMGI**). Without a damage initiation criterion, the damage evolution law has no effect on the material. The following table summarizes the compatible damage-initiation criteria with specific damage evolution laws:

TB,DGME , , , TBOPT option	Compatible TB,DMGI Input C1, C2, C3, C4	
	TBOPT value	C1, C2, C3, C4
1 or MPDG	1	1, 2, ..., 19
2 or CDM	1	4 (Hashin failure criteria only)

4.22.2.1. Predicting Post-Damage Degradation of Brittle Anisotropic Materials

Use the progressive damage model (**TB,DMGE**) to predict post-damage degradation of brittle anisotropic materials, typically in fiber-reinforced composites. The undamaged material must be linearly elastic. The onset of damage is determined via failure criteria (**TB,DMGI**). For more information, see [Failure Criteria](#) in the *Mechanical APDL Theory Reference*.

Following the onset of damage, material stiffness reduction occurs immediately. The constitutive relationship for a damage material is given as:

$$\tilde{\sigma} = [D] \varepsilon$$

$$\sigma = [D]_d \varepsilon$$

where:

$\tilde{\sigma}$ = effective Cauchy stress (stress measured in the undamaged domain)

σ = nominal Cauchy stress (effective stress averaged over the entire domain, including both damaged and undamaged domains)

ε = total elastic strain

$[D]$ = undamaged elasticity matrix

$[D]_d$ = damaged elasticity matrix

For a general orthotropic material, the damaged elastic matrix is defined as:

$$[D]_d = \begin{bmatrix} \frac{C_{11}}{(1-d_f)} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{21} & \frac{C_{22}}{(1-d_m)} & C_{23} & 0 & 0 & 0 \\ C_{31} & C_{32} & \frac{C_{33}}{(1-d_m)} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{C_{44}}{(1-d_s)} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{C_{55}}{(1-d_s)} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{C_{66}}{(1-d_s)} \end{bmatrix}^{-1}$$

where:

$[C]$ = compliance matrix of the undamaged material

d_f, d_m, d_s = fiber, matrix, and shear damage variables

Valid values are between 0 and 1, where

0 = no damage and 1 = complete loss of stiffness in the affected mode

For a transversely isotropic material with plane stress state, primarily adopted for thin fiber-reinforced composite structures, the damaged elastic matrix can be expressed as:

$$[D]_d = \frac{1}{A} \begin{bmatrix} (1-d_f)E_f & (1-d_f)(1-d_m)v_{21}E_f & 0 \\ (1-d_f)(1-d_m)v_{12}E_f & (1-d_m)E_m & 0 \\ 0 & 0 & A(1-d_s)G_{fm} \end{bmatrix}$$

where:

$$A = 1 - v_{12}v_{21}(1-d_f)(1-d_m)$$

E_f, E_m, G_{fm} = undamaged elastic and shear moduli

v_{12}, v_{21} = Poisson's ratios for the undamaged material

Four damage modes (p. 226) (fiber tension [rupture], fiber compression [kinking], matrix tension [cracking], and matrix compression [crushing]) are accounted for. Four damage variables (one for

each mode) are used to measure damage. The damage variables for calculating the damaged elasticity matrix are determined as follows:

$$d_f = \begin{cases} d_f^+, & \text{if } \lambda_f^+ > 0 \\ d_f^-, & \text{if } \lambda_f^- > 0 \end{cases}$$

$$d_m = \begin{cases} d_m^+, & \text{if } \lambda_m^+ > 0 \\ d_m^-, & \text{if } \lambda_m^- > 0 \end{cases}$$

$$ds = 1 - (1 - d_f^+)(1 - d_f^-)(1 - d_m^+)(1 - d_m^-)$$

where:

$$\lambda_f^+, \lambda_f^-, \lambda_m^+, \lambda_m^- = \text{fiber tension, fiber compression, matrix tension, matrix tension, and matrix compression failure calculated from effective stress } \tilde{\sigma}$$

When a damage mode is initiated, the damage progresses immediately, indicated by the increasing damage variable for the mode.

Two damage evolution methods are available:

- **Material property degradation method (TB,DMGE,,,,MPDG)**

The material stiffness is instantly reduced based on the damage variables, explicitly specified via the TB,DMGE command. Any physical failure criteria can be used to detect the onset of the damage.

- **Continuum damage mechanics method (TB,DMGE,,,,CDM)**

Damage variables increase gradually based on the energy amounts dissipated for the various damage modes. To achieve an objective response, the dissipated energy for each damage mode is regularized as follows:

$$g_v = \frac{g_c}{L_e}$$

where:

g_v = energy dissipated per unit volume

g_c = energy dissipated per unit area (fracture toughness, specified via TB,DMGE)

L_e = characteristic length of the element

The characteristic length L_e is calculated from the element area A via the following expressions:

$$L_e = \begin{cases} 1.12\sqrt{A}, & \text{for a square element} \\ 1.52\sqrt{A}, & \text{for a triangular element} \end{cases}$$

With this characteristic length, the constitutive relation is converted from stress-strain to stress-displacement relation. The damage evolution function is derived from Hashin failure criteria; therefore, only Hashin failure criteria are allowed via TB,DMGI for detecting damage onset. For

a plane stress state, the equivalent displacements $\|u\|$ and equivalent stresses $\|\sigma\|$ are given below for all damage modes.

4.22.2.1.1. Damage Modes

Four damage modes can be accounted for via the progressive damage model:

Fiber Tension Damage Mode

$$\|u\|_f^+ = L_e \sqrt{\langle \varepsilon_{ff} \rangle^2 + \alpha \varepsilon_{fm}^2}$$

$$\|\sigma\|_f^+ = \frac{L_e (\langle \sigma_{ff} \rangle \langle \varepsilon_{11} \rangle + \alpha \sigma_{fm} \varepsilon_{fm})}{\|u\|_f^+}$$

To ensure the continuity near the free fiber stress state ($\sigma_{ff} \approx 0$), the coupling factor α is chosen as follows:

$$\alpha = \begin{cases} 1, & \text{if } \sigma_{ff} \geq \sigma \\ \frac{\sigma_{ff}}{\sigma}, & \text{if } \sigma_{ff} < \sigma \end{cases}$$

where:

σ = a small tolerance value, selected based on the material fiber tensile strength

Fiber Compression Damage Mode

$$\|u\|_f^- = L_e \langle -\varepsilon_{ff} \rangle$$

$$\|\sigma\|_f^- = \langle -\sigma_{ff} \rangle$$

Matrix Tension Damage Mode

$$\|u\|_m^+ = L_e \sqrt{\langle \varepsilon_{mm} \rangle^2 + \varepsilon_{fm}^2}$$

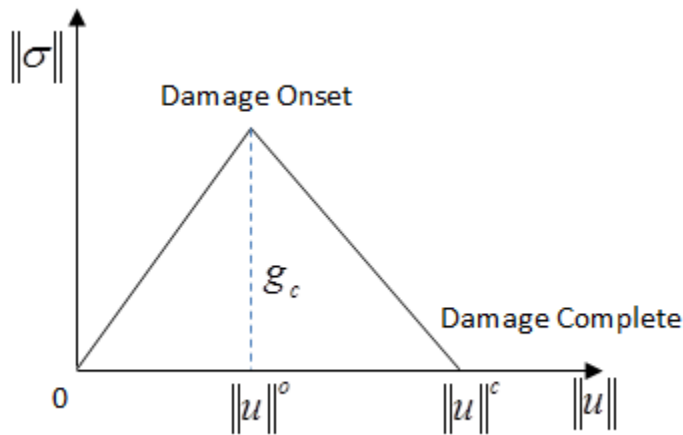
$$\|\sigma\|_m^+ = \frac{L_e (\langle \sigma_{mm} \rangle \langle \varepsilon_{mm} \rangle + \sigma_{fm} \varepsilon_{fm})}{\|u\|_m^+}$$

Matrix Compression Damage Mode

$$\|u\|_m^- = L_e \sqrt{\langle -\varepsilon_{mm} \rangle^2 + \varepsilon_{fm}^2}$$

$$\|\sigma\|_m^- = \frac{L_e (\langle -\sigma_{mm} \rangle \langle -\varepsilon_{mm} \rangle + \sigma_{fm} \varepsilon_{fm})}{\|u\|_m^-}$$

In expressions above, $\langle x \rangle$ is the McCauley operator, defined as $\langle x \rangle = \frac{x+|x|}{2}$.



The damage variable d for a given mode is given as follows:

$$d = \frac{||u||^f ||u||^0 - ||u||^0 ||u||^c}{||u||^f ||u||^c - ||u||^0 ||u||^c}$$

where:

$||u||^0$ = equivalent displacement at the damage onset

$||u||^f$ = equivalent displacement when the material reaches its ultimate strength

4.23. Material Damping

Damping can be introduced as a material property to account for dissipative mechanisms in dynamic analyses.

Contributions from the various material damping sources are additive. The contribution of material damping, in turn, is additive to the total structural damping. For a list of all available damping sources, see [Damping Matrices in the Theory Reference](#).

The following material damping topics are available:

[4.23.1. Material-Dependent Alpha and Beta Damping \(Rayleigh Damping\)](#)

[4.23.2. Material-Dependent Structural Damping](#)

[4.23.3. Viscoelastic Material Damping \(Harmonic Viscoelasticity\)](#)

4.23.1. Material-Dependent Alpha and Beta Damping (Rayleigh Damping)

Material-dependent damping enables you to specify mass proportional Rayleigh damping (alpha damping) or stiffness proportional Rayleigh damping (beta damping) as a material property.

Rayleigh damping is widely used to provide an energy-dissipation mechanism when analyzing complex engineering structures responding to dynamic loads such as seismic ground motion.

For more information about Rayleigh damping, see [Alpha and Beta Damping \(Rayleigh Damping\) in the Structural Analysis Guide](#).

4.23.1.1. Defining Material-Dependent Alpha and Beta Damping (Rayleigh Damping)

Define material-dependent alpha and beta damping using either linear material property (**MP**) or data-table (**TB**) commands. Both methods have advantages, as follows:

4.23.1.1.1. Defining Rayleigh Damping via MP Commands

4.23.1.1.2. Defining Rayleigh Damping via TB Commands

4.23.1.1.3. Comparing Feature Support: MP- vs. TB-Defined Damping

If Rayleigh damping is defined using **MP** and **TB** commands, the damping matrix is the sum of each source of damping as shown in Equation 14.22.

4.23.1.1.1. Defining Rayleigh Damping via MP Commands

Define mass proportional Rayleigh damping via **MP**,ALPD, and stiffness proportional Rayleigh damping via **MP**,BETD.

Rayleigh damping via **MP** commands defines the damping coefficients for the whole element, but not for each material in a multi-material element. For multi-material elements such as SOLID185 Layered Structural Solid, therefore, damping coefficients are determined from the material pointer for the element (**MAT**) rather than from the section materials.

Rayleigh damping defined via **MP** commands is not temperature-dependent and are always evaluated at $T = 0.0$.

To define Rayleigh damping, specify the damping option(s) and the constants (ALPD and/or BETD), as follows:

Constant	Meaning	Property	<i>Mechanical APDL Theory Reference Equation</i>
ALPD	α_i^m	Mass proportional material damping for material i	Equation 14.22
BETD	β_j^m	Stiffness proportional material damping for material j	Equation 14.22

Example 4.43: Defining Mass Proportional Material Damping (**MP**,ALPD)

```
/prep7
! Elastic properties and density
mp,ex,1,2.0e11
mp,nuxy,1,0.2
mp,dens,1,7800

! Mass proportional damping
mp,alpd,1,50
```

Stiffness proportional damping can be defined in a similar manner via **MP**,BETD.

4.23.1.1.2. Defining Rayleigh Damping via **TB** Commands

This method of defining damping coefficients is suitable for multi-material elements requiring definition of damping coefficients for section materials. Define mass proportional Rayleigh damping via **TB,SDAMP,,,ALPD**, and stiffness proportional Rayleigh damping via **TB,SDAMP,,,BETD**.

This method of defining damping parameters can be applicable to modeling layered composite materials consisting of a damped layer sandwiched between non-damped layers. It is also a powerful way of defining Rayleigh damping when field-variable dependence is desired.

This damping feature also allows damping coefficients to be specified for the element as a whole, as is done via Rayleigh-damping **MP** commands. If damping is uniform for the element with no field variable dependence, however, consider using the corresponding **MP** commands. When damping is defined via **TB** commands, the damping matrices are integrated similar to the structural stiffness and mass matrices and may be more resource-intensive.

The stiffness proportional damping matrix calculated when using **TB,SDAMP,,,BETD** does not include the stress stiffening included in the structural stiffness due to nonlinear geometry (**NLGEOM,ON**). Stress stiffening is ignored in the damping matrix calculation as it is not a material property. To include stress-stiffening in the damping matrix calculation, use **MP,BETD**.

When defining Rayleigh damping via **TB**:

1. Initialize the material data table (**TB,SDAMP,,,ALPD** and/or **TB,SDAMP,,,BETD**).
2. Enter the appropriate damping coefficient (**TBDATA**).

Constant	Meaning	Property	<i>Mechanical APDL Theory Reference Equation</i>
C1	α_p	Mass proportional material damping for material p	Equation 14.22
C1	β_q	Stiffness proportional material damping for material q	Equation 14.22

Both mass- and stiffness-proportional damping coefficients can be defined as frequency-, temperature-, or time-dependent properties (**TBFIELD**). Frequency-dependent properties are supported for full harmonic analyses only.

Example 4.44: Mass Proportional Material Damping (**TB,SDAMP,,,BETD**)

This example shows stiffness proportional damping as a function of temperature, with temperatures specified for each set of constants (**TBFIELD,TEMP**).

Separate damping constants are assigned to the layers by defining different stiffness proportional damping values for each of the layer materials. Mass proportional damping can be defined in a similar manner.

```
/prep7
! Elastic properties and density
mp,ex,1,2.0e11          ! Material 1
mp,nuxy,1,0.25
mp,dens,1,7800

mp,ex,2,2.2e11          ! Material 2
mp,nuxy,2,0.2
```

```

mp,dens,2,5800

! Stiffness proportional damping
TB,SDAMP,1,,,BETD      ! Material 1
TBFIELD, TEMP, 10      ! Define first temperature
TBDATA,,1.0e-3
TBFIELD, TEMP, 100     ! Define second temperature
TBDATA,,5.0e-3

TB,SDAMP,2,,,BETD      ! Material 2
TBFIELD, TEMP, 40      ! Define first temperature
TBDATA,,2.0e-3
TBFIELD, TEMP, 100     ! Define second temperature
TBDATA,,1.0e-3

! Apply section specific damping
sect,1,shell           ! Specify shell section
secd,0.1,1             ! Layer 1, Material 1
secd,0.1,2             ! Layer 2, Material 2
secd,0.1,1             ! Layer 3, Material 1

```

4.23.1.1.3. Comparing Feature Support: **MP**- vs. **TB**-Defined Damping

The following table summarizes the feature support for Rayleigh material damping defined using **MP** commands and **TB** commands:

Table 4.31: Support for Rayleigh Material Damping Defined using MP and TB Commands

Feature	MP ,ALPD	MP ,BETD	TB ,SDAMP,,,,ALPD	TB ,SDAMP,,,,BETD
Support for material-dependent damping	Y	Y	Y	Y
Support for different coefficients in multi-layer elements	--	--	Y	Y
Temperature dependence	--	--	Y	Y
Frequency dependence	--	--	Y	Y
Time dependence	--	--	Y	Y
Stress stiffening consideration in damping matrix	NA	Y	NA	--

4.23.2. Material-Dependent Structural Damping

Material structural damping is a form of stiffness proportional damping. This damping option allows you to specify the structural damping coefficient(s) m_j and/or g_j^E for each material j .

For more information, see [Harmonic Analysis in the Structural Analysis Guide](#).

You can define material-dependent structural damping using either linear material property (**MP**) or data-table (**TB**) commands:

4.23.2.1. Using **MP** to Define Material-Dependent Structural Damping Coefficients

4.23.2.2. Using **TB** to Define Material-Dependent Structural Damping Coefficients

4.23.2.3. Comparing Structural Damping Coefficient Options

4.23.2.1. Using **MP** to Define Material-Dependent Structural Damping Coefficients

You can define the structural damping coefficient m_j using **MP**,DMPS.

Constant	Meaning	Property	Mechanical APDL Theory Reference Equation
DMPS	m_j	Structural damping coefficient for material j	Equation 14.24

Example 4.45: Structural Damping (**MP**,DMPS)

```
/prep7
! Elastic properties and density
mp,ex,1,2.0e11
mp,nuxy,1,0.25
mp,dens,1,7800

! Structural damping
mp,dmps,1,0.1
```

4.23.2.2. Using **TB** to Define Material-Dependent Structural Damping Coefficients

You can define the structural damping coefficient g_j^E using **TB**,SDAMP,,,,STRU.

Defining structural damping using this method specifies damping in terms of the loss factor, equal to twice the damping ratio at the resonance frequency.

For the relationship between different structural damping coefficients, see [Comparing Structural Damping Coefficient Options](#) (p. 232).

Define the elastic behavior using the elasticity table (**TB**,ELASTIC), and density via the **MP** command. Initialize the material data table (**TB**,SDAMP,,,,STRU), and enter the appropriate damping coefficient (**TB**DATA).

Constant	Meaning	Property	Mechanical APDL Theory Reference Equation
C1	g_j^E	Structural damping coefficient for material j	Equation 14.24

For a full harmonic analysis, the structural damping coefficient can be frequency- or temperature-dependent (**TB**FIELD).

Example 4.46: Structural Damping (**TB**,SDAMP,,,,STRU)

```
/prep7
! Elastic properties and density
tb,elastic,1
tbdata,,2.0e11,0.25
mp,dens,1,7800

! Structural damping
tb,sdamp,1      ! TBOPT=STRU is the default
tbdata,,0.2
```

4.23.2.3. Comparing Structural Damping Coefficient Options

The relationship between **TB,SDAMP,,,STRU**, **DMPSTR**, **MP,DMPS**, and **MP,BETD** is as follows:

Table 4.32: Comparison: Structural Damping Coefficient Options

TB,SDAMP,,,STRU	DMPSTR	MP,DMPS	MP,BETD
s	s	s	$s / 2\pi f$

where s is the damping value specified in **TB,SDAMP,,,STRU** and f is the corresponding frequency.

4.23.3. Viscoelastic Material Damping (Harmonic Viscoelasticity)

The generalized Maxwell model consists of springs representing elastic stiffness and dashpots representing dissipative mechanisms inherent in viscoelastic materials. (See [Figure 4.22: Generalized Maxwell Solid in One Dimension \(p. 97\)](#).) The Maxwell model is useful for modeling damping mechanisms in harmonic analyses.

In the harmonic domain, the Maxwell model consists of imaginary components of the shear and bulk moduli ([Equation 4.36 \(p. 104\)](#)), commonly referred to as the loss moduli. The loss moduli are used to construct the viscoelastic damping matrix ($[K_m]$ in [Equation 14.24](#)). This is done by using the loss moduli divided by frequency and subsequently forming the damping matrix in a manner similar to the formation of the element stiffness matrix.

For more information, see [Harmonic Viscoelasticity \(p. 104\)](#).

Chapter 5: Multiphysics Material Properties

Material properties in multiphysics analyses can be linear (typically defined via the **MP** command) or nonlinear (defined using [data-table input \(p. 26\)](#) via the **TB** command).

The following topics for multiphysics material properties are available:

- 5.1. Acoustics
- 5.2. Fluids
- 5.3. Electricity and Magnetism
- 5.4. Migration Model
- 5.5. Thermal Properties

For more information, see the [Coupled-Field Analysis Guide](#), and [Elements for Multiphysics Analysis in the Element Reference](#).

5.1. Acoustics

The following topics about acoustic materials are available:

- 5.1.1. Perforated Media
- 5.1.2. Acoustic Frequency-Dependent Materials
- 5.1.3. Low Reduced Frequency (LRF) Model of Acoustic Viscous-Thermal Media
- 5.1.4. Diffusion Properties for Room Acoustics

5.1.1. Perforated Media

The following topics are available for modeling perforated media in acoustics analyses:

- 5.1.1.1. Equivalent Fluid Model of Perforated Media
- 5.1.1.2. Poroelastic Acoustic Material
- 5.1.1.3. Transfer Admittance Matrix Models

5.1.1.1. Equivalent Fluid Model of Perforated Media

The following equivalent fluid model topics are available:

- 5.1.1.1.1. Johnson-Champoux-Allard Equivalent Fluid Model of Perforated Media
- 5.1.1.1.2. Delany-Bazley Equivalent Fluid Model of Perforated Media
- 5.1.1.1.3. Miki Equivalent Fluid Model of Perforated Media
- 5.1.1.1.4. Complex Impedance and Propagating-Constant Equivalent Fluid Model of Perforated Media
- 5.1.1.1.5. Complex Density and Velocity Equivalent Fluid Model of Perforated Media

5.1.1.1.1. Johnson-Champoux-Allard Equivalent Fluid Model of Perforated Media

To define a Johnson-Champoux-Allard equivalent fluid model of a perforated medium in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,,JCA

The effective density is given by:

$$\rho = \frac{\rho_0 \alpha_\infty}{\varphi} \left[1 + \frac{\sigma \varphi}{j \omega \rho_0 \alpha_\infty} \left(1 + \frac{j 4 \alpha_\infty^2 \eta \omega \rho_0}{\varphi^2 \Lambda^2 \sigma^2} \right)^{1/2} \right] \quad (5.1)$$

where:

ρ_0 = Fluid density

σ = Fluid resistivity

φ = Porosity

α_∞ = Tortuosity

Λ = Viscous characteristic length

η = Dynamic viscosity

The effective bulk modulus is given by:

$$K(\omega) = \frac{P_0 \gamma / \varphi}{\gamma - (\gamma - 1) \left[\frac{8 \eta}{j \omega \rho_0 P_{rt} \Lambda'^2} \left(1 + \frac{\Lambda'^2}{16} \frac{j \omega \rho_0 P_{rt}}{\eta} \right)^{1/2} + 1 \right]^{-1}} \quad (5.2)$$

where:

γ = Specific heat ratio

P_0 = Static reference pressure

P_{rt} = Prandtl number

Λ' = Thermal characteristic length

The constants C1 through C5 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Fluid resistivity (N·s/m ⁴)
C2	Porosity (defaults to 1)
C3	Tortuosity (defaults to 1)
C4	Viscous characteristic length (m)
C5	Thermal characteristic length (m)

Additional material parameters are input via the **MP** and **R** commands. For more information, see [Equivalent Fluid Model of Perforated Material in the Acoustic Analysis Guide](#) and [Equivalent Fluid of Perforated Materials in the Theory Reference](#).

5.1.1.1.2. Delany-Bazley Equivalent Fluid Model of Perforated Media

To define a Delany-Bazley equivalent fluid model of a perforated medium in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,DLB

The impedance is given by:

$$Z_c = R + jX$$

The propagating constant is given by:

$$\gamma = \alpha + j\beta$$

where:

$$R = \rho_0 c_0 \left\{ 1 + 0.0497 \left(\frac{f}{\sigma} \right)^{-0.754} \right\}$$

$$X = -\rho_0 c_0 \left\{ 0.0758 \left(\frac{f}{\sigma} \right)^{-0.732} \right\}$$

$$\alpha = \frac{\omega}{c_0} 0.169 \left(\frac{f}{\sigma} \right)^{-0.618}$$

$$\beta = \frac{\omega}{c_0} \left\{ 1 + 0.0858 \left(\frac{f}{\sigma} \right)^{-0.700} \right\}$$

σ = Fluid resistivity

f = Frequency

ω = Angular frequency

c_0 = speed of sound

The constant C1 (entered via the **TBDATA** command) is:

Constant	Meaning
C1	Fluid resistivity (N·s/m ⁴)

5.1.1.1.3. Miki Equivalent Fluid Model of Perforated Media

To define a Miki equivalent fluid model of a porous medium in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,MIKI

The impedance is given by:

$$Z_c = R + jX$$

The propagating constant is given by:

$$\gamma = \alpha + j\beta$$

where:

$$R = \rho_0 c_0 \left\{ 1 + 0.0699 \left(\frac{f}{\sigma} \right)^{-0.632} \right\}$$

$$X = -\rho_0 c_0 \left\{ 0.107 \left(\frac{f}{\sigma} \right)^{-0.632} \right\}$$

$$\alpha = \frac{\omega}{c_0} 0.160 \left(\frac{f}{\sigma} \right)^{-0.618}$$

$$\beta = \frac{\omega}{c_0} \left\{ 1 + 0.109 \left(\frac{f}{\sigma} \right)^{-0.618} \right\}$$

σ = Fluid resistivity

f = Frequency

ω = Angular frequency

The constant C1 (entered via the **TBDATA** command) is:

Constant	Meaning
C1	Fluid resistivity (N·s/m ⁴)

5.1.1.1.4. Complex Impedance and Propagating-Constant Equivalent Fluid Model of Perforated Media

To define a complex impedance and propagating-constant equivalent fluid model of a porous medium in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,ZPRO

The impedance is given by:

$$Z_c = R + jX$$

The propagating constant is given by:

$$\gamma = \alpha + j\beta$$

where:

R = Resistance

X = Reactance

α = Attenuation constant

β = Phase constant

The constants C1 through C4 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Resistance (Pa·s /m)
C2	Reactance (Pa·s /m)
C3	Attenuation constant (Nepers/m)

Constant	Meaning
C4	Phase constant (Rad/m)

5.1.1.1.5. Complex Density and Velocity Equivalent Fluid Model of Perforated Media

To define a complex impedance-propagating constant equivalent fluid model of a porous medium in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,CDV

The complex density is given by:

$$\rho = \rho_r + j\rho_i$$

The complex sound speed is given by:

$$c = c_r + jc_i$$

where:

ρ_r = Real part of complex density (kg/m³)

ρ_i = Imaginary part of complex density (kg/m³)

c_r = Real part of complex sound speed (m/s)

c_i = Imaginary part of complex sound speed (m/s)

The constants C1 through C4 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Real part of complex density (kg/m ³)
C2	Imaginary part of complex density (kg/m ³)
C3	Real part of complex sound speed (m/s)
C4	Imaginary part of complex sound speed (m/s)

5.1.1.2. Poroelastic Acoustic Material

To define a poroelastic acoustic material of a perforated medium in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,PORO

The constants C1 through C10 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Fluid resistivity (N·s/m ⁴)
C2	Porosity (defaults to 1)
C3	Tortuosity (defaults to 1)

Constant	Meaning
C4	Viscous characteristic length (m)
C5	Thermal characteristic length (m)
C6	Bulk density of solid phase of the poroelastic material (kg/m ³)
C7	Loss factor of elasticity moduli (defaults to 0)
C8	Loss factor of shear moduli (defaults to 0)
C9	Biot's coefficient (defaults to 1)
C10	Bulk modulus of the elastic solid from which the frame is made (N/m ²) (defaults to 0)

Additional material parameters are input via the **MP** command. For more information, see [Poroelastic Acoustic Material in the Acoustic Analysis Guide](#) and [Poroelastic Acoustics in the Theory Reference](#).

5.1.1.3. Transfer Admittance Matrix Models

The following transfer admittance matrix topics are available:

[5.1.1.3.1. Transfer Admittance Matrix Model of Perforated Media](#)

[5.1.1.3.2. Transfer Admittance Matrix Model of a Square or Hexagonal Grid Structure](#)

5.1.1.3.1. Transfer Admittance Matrix Model of Perforated Media

To define a transfer admittance matrix model of a porous medium in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,YMAT

A two-port transfer admittance matrix is given by:

$$\begin{Bmatrix} v_{n1} \\ v_{n2} \end{Bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} \begin{Bmatrix} p_1 \\ p_2 \end{Bmatrix} + \begin{Bmatrix} \alpha_1 \\ \alpha_2 \end{Bmatrix}$$

where:

v_{n1} = Normal velocity at port 1

p_1 = Pressure at port 1

v_{n2} = Normal velocity at port 2

p_2 = Pressure at port 2

$Y_{11}, Y_{12}, Y_{21}, Y_{22}$ = Complex admittance elements

α_1 = Internal source related to port 1 (usually zero in acoustic applications)

α_2 = Internal source related to port 2 (usually zero in acoustic applications)

The constants C1 through C12 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Real part of complex Y_{11} (m/Pa·s)
C2	Imaginary part of complex Y_{11} (m/Pa·s)
C3	Real part of complex Y_{12} (m/Pa·s)
C4	Imaginary part of complex Y_{12} (m/Pa·s)
C5	Real part of complex Y_{21} (m/Pa·s)
C6	Imaginary part of complex Y_{21} (m/Pa·s)
C7	Real part of complex Y_{22} (m/Pa·s)
C8	Imaginary part of complex Y_{22} (m/Pa·s)
C9	Real part of complex α_1 (m/s)
C10	Imaginary part of complex α_1 (m/s)
C11	Real part of complex α_2 (m/s)
C12	Imaginary part of complex α_2 (m/s)

For an acoustic 2 x 2 transfer admittance matrix, the port number (**SF,Nlist,PORT**) can be any positive integer.

If the two ports of the transfer admittance matrix are connecting to the fluid, the smaller port number corresponds to port 1 of the 2 x 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 and another port is connecting to the fluid, the FSI interface (**SF,Nlist,FSI**) corresponds to port 1 and the defined port number (**SF,Nlist,PORT**) corresponds to port 2 of the transfer admittance matrix.

A pair of ports of the 2 x 2 transfer admittance matrix must be defined in the same element.

5.1.1.3.2. Transfer Admittance Matrix Model of a Square or Hexagonal Grid Structure

To define a transfer admittance matrix model of a square grid structure in an acoustic full harmonic analysis, issue this command:

TB,PERF,,,SGYM

To define a hexagonal grid structure, issue this command:

TB,PERF,,,HGYM

A two-port transfer admittance matrix is given by:

$$\begin{Bmatrix} v_{n1} \\ v_{n2} \end{Bmatrix} = \begin{bmatrix} Y & -Y \\ -\beta Y & \beta Y \end{bmatrix} \begin{Bmatrix} p_1 \\ p_2 \end{Bmatrix}$$

where:

Y = Complex admittance elements determined by geometric dimension and material

β = Ratio of inner and outer radius for cylindrical structure (default = 1)

The constants C1 through C6 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Radius of the hole (m)
C2	Period of the square or hexagonal grid structure (m)
C3	Thickness of the structure (m)
C4	Mass density of fluid (kg/m ³)
C5	Dynamic viscosity of fluid (Pa·s)
C6	Ratio of inner and outer radius for cylindrical structure

5.1.2. Acoustic Frequency-Dependent Materials

To define frequency-dependent material in an acoustic full harmonic analysis, issue this command:

TB,AFDM,,,,MAT

The constants C1 through C7 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Mass density (kg/m ³)
C2	Sound speed (m/s)
C3	Dynamic viscosity (Pa·s)
C4	Thermal conductivity (W/m·K)
C5	Specific heat (J/kg·K)
C6	Heat coefficient at constant volume per unit of mass (J/kg·K)
C7	Bulk viscosity (Pa·s)

5.1.3. Low Reduced Frequency (LRF) Model of Acoustic Viscous-Thermal Media

The low reduced frequency (LRF) model is available for three cases:

5.1.3.1. Thin Layer

5.1.3.2. Tube with Rectangular Cross-Section

5.1.3.3. Tube with Circular Cross-Section

5.1.3.1. Thin Layer

To define the low reduced frequency model in an acoustic full harmonic analysis for a thin layer, issue this command:

TB,AFDM,,,,THIN

The constant C1 (entered via the **TBDATA** command) is:

Constant	Meaning
C1	Thickness of the layer

5.1.3.2. Tube with Rectangular Cross-Section

To define the low reduced frequency model in an acoustic full harmonic analysis for a tube with a rectangular cross-section, issue this command:

TB,AFDM,,,RECT

The constants C1 through C2 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Width of the rectangular cross-section
C2	Height of the rectangular cross-section

5.1.3.3. Tube with Circular Cross-Section

To define the low reduced frequency model in an acoustic full harmonic analysis for a tube with a circular cross-section, issue this command:

TB,AFDM,,,CIRC

The constant C1 (entered via the **TBDATA** command) is:

Constant	Meaning
C1	Radius of the circular cross-section

5.1.4. Diffusion Properties for Room Acoustics

To define frequency-dependent diffusion properties for room acoustics, issue this command:

TB,AFDM,,,ROOM

The constants C1 through C4 (entered via the **TBDATA** command) are:

Constant	Meaning
C1	Empty room diffusion coefficient (m^2/s)
C2	The coefficient of atmospheric attenuation ($1/\text{m}$)
C3	Furniture diffusion coefficient (m^2/s)
C4	Furniture absorption coefficient

5.2. Fluids

Fluid material models can be used with hydrostatic fluid elements to model compressible fluids. For theoretical background on these materials, see [Fluid Material Models](#) in the *Mechanical APDL Theory Reference*. For more information on using these fluid material models with the hydrostatic fluid elements, see [Modeling Hydrostatic Fluids](#) in the *Structural Analysis Guide*.

There are three ways to define material data for compressible fluids: liquid, gas, or pressure-volume data.

Liquid

Use the **TB,FLUID** command with *TBOPT* = LIQUID to define material behavior for a liquid, and specify the following material constants using the **TBDATA** command:

Constant	Meaning	Property
C1	K	Bulk modulus
C2	α	Coefficient of thermal expansion
C3	ρ_{0f}	Initial density

You can define a temperature dependent liquid material with up to 20 temperatures (*NTEMP* = 20 max on the **TB** command) by using the **TBTEMP** command, as shown in the example below:

```
TB,FLUID,1,2,,LIQUID ! Activate liquid material model
TBTEMP,100.0         ! Define first temperature
TBDATA,1,K, , 0f      ! Define material constants at temp 100.0
TBTEMP,200.0         ! Define second temperature
TBDATA,1,K, , 0f      ! Define material constants at temp 200.0
```

When specifying temperature dependent density values for a liquid, consider that the current density (ρ_f) for hydrostatic fluid elements is computed at each iteration as a function of pressure change (ΔP), bulk modulus (K), coefficient of thermal expansion (α), and temperature change (ΔT). A reference temperature may be input using the **TREF** or **MP,REFT** command. For details on how the current density is calculated, refer to [Liquid](#) in the *Mechanical APDL Theory Reference*.

Gas

Use the **TB,FLUID** command with *TBOPT* = GAS to define material behavior for a gas, and specify the following material constant using the **TBDATA** command:

Constant	Meaning	Property
C1	ρ_{0f}	Initial density

You can define a temperature dependent gas material with up to 20 temperatures (*NTEMP* = 20 max on the **TB** command) by using the **TBTEMP** command, as shown in the example below:

```
TB,FLUID,1,2,,GAS    ! Activate gas material model
TBTEMP,100.0         ! Define first temperature
TBDATA,1, 0f         ! Define material constants at temp 100.0
TBTEMP,200.0         ! Define second temperature
TBDATA,1, 0f         ! Define material constants at temp 200.0
```

When specifying temperature dependent density values for a gas, consider that the current density (ρ_f) for hydrostatic fluid elements is computed at each iteration based on the Ideal Gas Law. For details on how the current density is calculated, refer to [Gas](#) in the *Mechanical APDL Theory Reference*.

To use the Ideal Gas Law, you also need to define a reference pressure (input as real constant PREF) and a reference temperature (input with the **TREF** or **MP,REFT** command) with temperature offset (input with the **TOFFST** command).

Pressure-Volume Data

Use the **TB,FLUID** command with **TBOPT** = PVDATA to define compressible fluid behavior in terms of a pressure-volume curve. You can specify up to 20 temperature-dependent pressure-volume curves (**NTEMP** = 20 max on the **TB** command). The temperature for the first curve is input with **TBTEMP**, followed by **TBPT** commands for up to 100 pressure-volume data points. The data points (X, Y) entered on **TBPT** are:

Constant	Meaning
X	Pressure value
Y	Corresponding volume value

The pressure-volume data point must be defined in terms of total pressure and total volume of the fluid in the containing vessel.

5.3. Electricity and Magnetism

The following material model topics related to electricity and magnetism are available:

5.3.1. Piezoelectricity

5.3.2. Piezoresistivity

5.3.3. Magnetism

5.3.4. Anisotropic Electric Permittivity

5.3.5. Anisotropic Viscosity

5.3.6. Anisotropic Elastic Loss Tangent

5.3.7. Anisotropic Dielectric Loss Tangent

5.3.1. Piezoelectricity

Piezoelectric capability (**TB,PIEZ**) is available with the coupled-field elements. (See [Material Model Element Support](#) (p. 5) for piezoelectricity.) Material properties required for the piezoelectric effects include the dielectric (relative permittivity) constants, the elastic coefficient matrix, and the piezoelectric matrix.

You can define the piezoelectric stress matrix at constant strain [e] (**TBOPT** = 0) or the piezoelectric strain matrix [d] (**TBOPT** = 1). The [e] matrix is typically associated with the input of the anisotropic elasticity in the form of the stiffness matrix [c], and the permittivity at constant strain [ϵ^S]. The [d] matrix is associated with the input of compliance matrix [s] and permittivity at constant stress [ϵ^T]. Select the appropriate matrix form for your analysis using the **TB,PIEZ** command.

The full 6 x 3 piezoelectric matrix relates terms x, y, z, xy, yz, xz to x, y, z via 18 constants as shown:

$$\begin{bmatrix} e_{11} & e_{12} & e_{13} \\ e_{21} & e_{22} & e_{23} \\ e_{31} & e_{32} & e_{33} \\ e_{41} & e_{42} & e_{43} \\ e_{51} & e_{52} & e_{53} \\ e_{61} & e_{62} & e_{63} \end{bmatrix}$$

For 2-D problems, a 4 x 2 matrix relates terms ordered x, y, z, xy via 8 constants ($e_{11}, e_{12}, e_{21}, e_{22}, e_{31}, e_{32}, e_{41}, e_{42}$). The order of the vector is expected as $\{x, y, z, xy, yz, xz\}$, whereas for some published materials the order is given as $\{x, y, z, yz, xz, xy\}$. This difference requires the piezoelectric matrix terms to be converted to the expected format.

You can define up to 18 constants (C1-C18) with **TBDATA** commands (6 per command):

Constant	Meaning
C1-C6	Terms $e_{11}, e_{12}, e_{13}, e_{21}, e_{22}, e_{23}$
C7-C12	Terms $e_{31}, e_{32}, e_{33}, e_{41}, e_{42}, e_{43}$
C13-C18	Terms $e_{51}, e_{52}, e_{53}, e_{61}, e_{62}, e_{63}$

In a piezoelectric analysis using **PLANE223**, **SOLID226**, or **SOLID227**, you can define temperature-dependent piezoelectric coefficients (with **TBOPT** = 0) by using tabular input: **TBDATA,STLOC,%tabname%**, where *tabname* is the name of a table array parameter (***DIM**) with TEMP as a primary variable. For more information, see [Defining Linear Material Properties Using Tabular Input \(p. 16\)](#).

See [Piezoelectric Analysis](#) in the *Coupled-Field Analysis Guide* for more information about this material model.

5.3.2. Piezoresistivity

Elements with piezoresistive capabilities use the **TB,PZRS** command to calculate the change in electric resistivity produced by elastic stress or strain. Material properties required to model piezoresistive materials are electrical resistivity, the elastic coefficient matrix, and the piezoresistive matrix.

You can define the piezoresistive matrix either in the form of piezoresistive stress matrix $[\pi]$ (**TBOPT** = 0) or piezoresistive strain matrix $[m]$ (**TBOPT** = 1).

The piezoresistive stress matrix $[\pi]$ uses stress to calculate the change in electric resistivity due to piezoresistive effect, while the piezoresistive strain matrix $[m]$ (**TBOPT** = 1) uses strain to calculate the change in electric resistivity. See [Piezoresistivity](#) in the *Mechanical APDL Theory Reference* for more information.

The full 6x6 piezoresistive matrix relates the x, y, z, xy, yz, xz terms of stress to the x, y, z, xy, yz, xz terms of electric resistivity via 36 constants:

$$\begin{bmatrix} \pi_{11} & \pi_{12} & \pi_{13} & \pi_{14} & \pi_{15} & \pi_{16} \\ \pi_{21} & \pi_{22} & \pi_{23} & \pi_{24} & \pi_{25} & \pi_{26} \\ \pi_{31} & \pi_{32} & \pi_{33} & \pi_{34} & \pi_{35} & \pi_{36} \\ \pi_{41} & \pi_{42} & \pi_{43} & \pi_{44} & \pi_{45} & \pi_{46} \\ \pi_{51} & \pi_{52} & \pi_{53} & \pi_{54} & \pi_{55} & \pi_{56} \\ \pi_{61} & \pi_{62} & \pi_{63} & \pi_{64} & \pi_{65} & \pi_{66} \end{bmatrix}$$

Constant	Meaning
C1-C6	Terms $\pi_{11}, \pi_{12}, \pi_{13}, \pi_{14}, \pi_{15}, \pi_{16}$
C7-C12	Terms $\pi_{21}, \pi_{22}, \pi_{23}, \pi_{24}, \pi_{25}, \pi_{26}$
C13-C18	Terms $\pi_{31}, \pi_{32}, \pi_{33}, \pi_{34}, \pi_{35}, \pi_{36}$
C19-C24	Terms $\pi_{41}, \pi_{42}, \pi_{43}, \pi_{44}, \pi_{45}, \pi_{46}$
C25-C30	Terms $\pi_{51}, \pi_{52}, \pi_{53}, \pi_{54}, \pi_{55}, \pi_{56}$
C31-C36	Terms $\pi_{61}, \pi_{62}, \pi_{63}, \pi_{64}, \pi_{65}, \pi_{66}$

For 2-D problems, a 4x4 matrix relates terms ordered x, y, z, xy via 16 constants.

Constant	Meaning
C1-C4	Terms $\pi_{11}, \pi_{12}, \pi_{13}, \pi_{14}$
C7-C10	Terms $\pi_{21}, \pi_{22}, \pi_{23}, \pi_{24}$
C13-C16	Terms $\pi_{31}, \pi_{32}, \pi_{33}, \pi_{34}$
C19-C22	Terms $\pi_{41}, \pi_{42}, \pi_{43}, \pi_{44}$

The order of the vector is expected as {x, y, z, xy, yz, xz}, whereas for some published materials the order is given as {x, y, z, yz, xz, xy}. This difference requires the piezoresistive matrix terms to be converted to the expected format.

See [Piezoresistive Analysis](#) in the *Coupled-Field Analysis Guide* for more information on this material model.

5.3.3. Magnetism

Elements with magnetic capability use the **TB** table to input points characterizing B-H curves.

Initialize the curves with the **TB,BH** command. Use **TBPT** commands to define up to 500 points (H, B). The constants (X, Y) entered on **TBPT** (two per command) are:

Constant	Meaning	Property
X	Magnetomotive force/length	Magnetic field intensity (H)
Y	Flux/Area	Corresponding magnetic flux density (B)

Specify the system of units (MKS or user defined) with **EMUNIT**, which also determines the value of the permeability of free space. This value is used with the relative permeability property values (**MP**) to establish absolute permeability values. The defaults (also obtained for *Lab* = MKS) are MKS units and free-space permeability of $4 \pi \text{E-}7$ Henries/meter. You can specify *Lab* = MUZRO to define any system of units, then input free-space permeability.

Inputting multiple B-H curves to define temperature dependency is not supported.

For soft magnetic materials, temperature dependency of the B-H curve can be modeled by setting $TBOPT = TCF$ on the **TB,BH** command. Use **TBTEMP** (or **TBFIELD,TEMP**) and **TBDATA** commands to define the thermal coefficient vs. temperature. The thermal coefficient is used to modify the B-H curve:

Constant	Meaning	Input Format
C1	Thermal Coefficient	TB, BH, MATID, , , TCF TBTEMP, TempValue (or TBFIELD, TEMP, TempValue) TBDATA, 1, C1

(repeat **TBTEMP** and **TBDATA** for each temperature point)

The C1 value should be within the range of 0 to 1.

This method of defining temperature dependency for B-H curves is not valid for permanent magnets.

For more information about this material option, see [Additional Guidelines for Defining Regional Material Properties and Real Constants](#) in the *Low-Frequency Electromagnetic Analysis Guide*

5.3.4. Anisotropic Electric Permittivity

Current technology coupled-field elements with piezoelectric and electrostatic-structural capabilities use the **TB,DPER** table to input the matrix of relative electric permittivity. (See [Material Model Element Support \(p. 5\)](#) for Anisotropic Electric Permittivity.) Alternatively, you can use the **MP** command to input electric permittivity as constants PERX, PERY, and PERZ.

The full 3 x 3 electric permittivity matrix:

$$[\epsilon] = \begin{bmatrix} \epsilon_{11} & \epsilon_{12} & \epsilon_{13} \\ & \epsilon_{22} & \epsilon_{23} \\ & & \epsilon_{33} \end{bmatrix}$$

relates x, y, z components of electric field intensity {E} to the x, y, z components of electric flux density {D} via 6 constants. For 2-D problems, a 2x2 matrix relates terms ordered x, y via three constants ($\epsilon_{11} \ \epsilon_{22} \ \epsilon_{12}$). The program will make the $[\epsilon]$ matrix symmetric.

You can define electric permittivity at constant strain $[\epsilon^S]$ ($TBOPT = 0$) or constant stress $[\epsilon^T]$ ($TBOPT = 1$)

The program converts matrix $[\epsilon^T]$ to $[\epsilon^S]$ using piezoelectric strain and stress matrices.

Define the permittivity matrix $[\epsilon]$ via the **TB** family of commands, as follows:

1. Initialize the constant table (**TB,DPER**). Set the appropriate $TBOPT$ value to input electric permittivity at constant strain or electric permittivity at constant stress.
2. Specify up to 6 constants (**TBDATA**).

3-D Input

Constant	Meaning
C1-C6	$\epsilon_{11}, \epsilon_{22}, \epsilon_{33}, \epsilon_{12}, \epsilon_{23}, \epsilon_{13}$

2-D Input

Constant	Meaning
C1, C2, C4	$\epsilon_{11}, \epsilon_{22}, \epsilon_{12}$

In a piezoelectric analysis using [PLANE223](#), [SOLID226](#), or [SOLID227](#), you can define temperature-dependent permittivity coefficients (with *TBOPT* = 0) by using tabular input: **TB***DATA,STLOC,%tabname%*, where *tabname* is the name of a table array parameter (***DIM**) with TEMP as a primary variable. For more information, see [Defining Linear Material Properties Using Tabular Input \(p. 16\)](#).

5.3.5. Anisotropic Viscosity

Current-technology coupled-field elements with piezoelectric capability use the **TB,AVIS** table to input the matrix of viscosity coefficients. Anisotropic viscosity is used in a transient or harmonic piezoelectric analysis to account for anisotropic viscous damping. For a list of elements that use this model, see [Material Model Element Support \(p. 5\)](#). For constitutive relations involving anisotropic viscosity, see [Piezoelectrics in the Theory Reference](#). For more information on using **TB,AVIS** in a piezoelectric analysis, see [Piezoelectric Analysis in the Coupled-Field Analysis Guide](#).

The full 6 x 6 viscosity coefficient matrix:

$$[\eta] = \begin{bmatrix} \eta_{11} & & & & & \\ \eta_{21} & \eta_{22} & & & & \\ \eta_{31} & \eta_{32} & \eta_{33} & & & \\ \eta_{41} & \eta_{42} & \eta_{43} & \eta_{44} & & \\ \eta_{51} & \eta_{52} & \eta_{53} & \eta_{54} & \eta_{55} & \\ \eta_{61} & \eta_{62} & \eta_{63} & \eta_{64} & \eta_{65} & \eta_{66} \end{bmatrix}$$

relates the stress $\{T\} = \{T_1, T_2, T_3, T_4, T_5, T_6\}^T$ and strain rate $\left\{\frac{\partial S}{\partial t}\right\} = \{\dot{S}_1, \dot{S}_2, \dot{S}_3, \dot{S}_4, \dot{S}_5, \dot{S}_6\}^T$ vector components.

The order of the vector components is expected to be {x, y, z, xy, yz, xz}. That is:

$$\{T\} = \{T_x, T_y, T_z, T_{xy}, T_{yz}, T_{xz}\}^T \text{ and } \left\{\frac{\partial S}{\partial t}\right\} = \{\dot{S}_x, \dot{S}_y, \dot{S}_z, \dot{S}_{xy}, \dot{S}_{yz}, \dot{S}_{xz}\}^T,$$

whereas for some published materials the order is given as {x, y, z, yz, xz, xy}. This difference requires the $[\eta]$ matrix to be converted to the expected format. Use the **TB***DATA* command to input up to 21 constants, forming the lower or upper triangle of the symmetric matrix $[\eta]$. For 2-D problems, a 4 x 4 matrix relates the stress and elastic strain rate vector components ordered {x, y, z, xy} via 10 constants ($\eta_{11}, \eta_{21}, \eta_{31}, \eta_{41}, \eta_{22}, \eta_{32}, \eta_{42}, \eta_{33}, \eta_{43}, \eta_{44}$). Using this input, the program makes the viscosity matrix $[\eta]$ symmetric. The resulting symmetric matrix should be positive semidefinite.

The **TB,AVIS** input can be either in the form of viscosity matrix $[\eta]$ (*TBOPT* = 0) or fluency matrix $[\zeta]$ (*TBOPT* = 1). Both forms use the same data table input. The viscosity matrix $[\eta]$ is typically input together with the stiffness matrix $[c]$ (**TB,ANEL** or **TB,ELAS,,,AELS**), while the fluency matrix $[\zeta]$ is commonly used together with the compliance matrix $[s]$ (**TB,ANEL**). If the fluency option is selected with

the **TB,AVIS** command, the program converts the fluency matrix $[\zeta]$ to viscosity matrix $[\eta]$ using the stiffness or compliance matrix input on **TB,ANEL**. A zero-frequency limit is assumed during the fluency to viscosity conversion.

Define the viscosity coefficient matrix $[\eta]$ via the **TB** family of commands, as follows:

1. Initialize the constant table (**TB,AVIS**). Set the appropriate *TBOPT* value to specify the viscosity or fluency form.
2. Specify up to 21 constants (**TB,DATA**).

3-D Input

Constant	Meaning
C1-C6	Terms $\eta_{11}, \eta_{21}, \eta_{31}, \eta_{41}, \eta_{51}, \eta_{61}$
C7-C12	Terms $\eta_{22}, \eta_{32}, \eta_{42}, \eta_{52}, \eta_{62}, \eta_{33}$
C13-C18	Terms $\eta_{43}, \eta_{53}, \eta_{63}, \eta_{44}, \eta_{54}, \eta_{64}$
C19-C21	Terms $\eta_{55}, \eta_{65}, \eta_{66}$

2-D Input

Constant	Meaning
C1-C4	Terms $\eta_{11}, \eta_{21}, \eta_{31}, \eta_{41}$
C7-C9	Terms $\eta_{22}, \eta_{32}, \eta_{42}$
C12-C13	Terms η_{33}, η_{43}
C16	Term η_{44}

Example 5.1: Defining Anisotropic Viscosity (**TB,AVIS**)

```
! Viscosity coefficients for Lithium Niobate (N/m**2 s)
et11= 0.6547e-3
et12= 0.2275e-3
et13= 0.2499e-3
et14=-0.0687e-3
et33= 0.3377e-3
et44= 0.1765e-3
et66= (et11-et12)/2

! Viscosity matrix in IEEE format
!
! [et11 et12 et13 et14 0 0 ]
! [et12 et11 et13 -et14 0 0 ]
! [et13 et13 et33 0 0 0 ]
! [et14 -et14 0 et44 0 0 ]
! [ 0 0 0 0 et44 et14]
! [ 0 0 0 0 et14 et66]
!
! Viscosity matrix in ANSYS format
!
! [et11 et12 et13 0 et14 0 ]
! [et12 et11 et13 0 -et14 0 ]
! [et13 et13 et33 0 0 0 ]
! [ 0 0 0 et66 0 et14]
! [et14 -et14 0 0 et44 0 ]
! [ 0 0 0 et14 0 et44]
!
! Anisotropic viscosity table input
tb,AVIS,1,,0
tbda,1,et11,et12,et13,,et14
tbda,7,et11,et13,, -et14
```

```
tbda,12,et33
tbda,16,et66,,et14
tbda,19,et44
tbda,21,et44
```

Reference

Viscosity coefficients for Lithium Niobate (N/m**2 s) are referenced from:

I.L Bajak, A. McNab, J. Richter, C.D.W. Wilkinson. "Attenuation of acoustic waves in lithium niobate". *The Journal of the Acoustical Society of America*. vol. 69, no. 3, pp. 689-95 (1981).

5.3.6. Anisotropic Elastic Loss Tangent

Current-technology coupled-field elements with piezoelectric capability use the **TB,ELST** table to input the matrix of elastic loss tangents (factors). Anisotropic elastic loss tangent is used in a harmonic piezoelectric analysis to account for anisotropic hysteretic damping. For elements that use this model, see [Material Model Element Support \(p. 5\)](#). For constitutive relations involving anisotropic elastic loss factors, see [Piezoelectrics in the Theory Reference](#). For more information on using **TB,ELST** in a piezoelectric analysis, see [Piezoelectric Analysis in the Coupled-Field Analysis Guide](#). Isotropic elastic loss tangent (structural damping coefficient) can be input using the **MP,DMP5** and **TB,SDAMP,,,,STRU** commands.

The full 6 x 6 elastic loss tangent matrix :

$$[\psi] = \begin{bmatrix} \psi_{11} & & & & & \\ \psi_{21} & \psi_{22} & & & & \\ \psi_{31} & \psi_{32} & \psi_{33} & & & \\ \psi_{41} & \psi_{42} & \psi_{43} & \psi_{44} & & \\ \psi_{51} & \psi_{52} & \psi_{53} & \psi_{54} & \psi_{55} & \\ \psi_{61} & \psi_{62} & \psi_{63} & \psi_{64} & \psi_{65} & \psi_{66} \end{bmatrix}$$

is used together with the 6 x 6 elastic stiffness matrix $[c]$ to form the imaginary part of the complex stiffness matrix $[c^*] = [c] + j[\psi] \cdot [c]$ in a harmonic piezoelectric analysis. The elastic stiffness matrix $[c]$ can be defined by either specifying the stiffness constants (EX, EY, etc.) with **MP** commands or by specifying the terms of the elasticity matrix via the **TB,ANEL** or **TB,ELAS** data table. The same elastic loss tangent matrix $[\psi]$ applies to the elastic stiffness $[c]$ and elastic flexibility $[s]$ forms.

The order of rows and columns in matrix $[\psi]$ corresponds to the stress or strain vector components ordered {x, y, z, xy, yz, xz}, whereas for some published materials the order is given as {x, y, z, yz, xz, xy}. This difference requires the $[\psi]$ matrix terms to be converted to the expected format. Use the **TB,DATA** command to input up to 21 constants forming the lower or upper triangle of symmetric matrix $[\psi]$. For 2-D problems, a 4 x 4 matrix is input using 10 constants ($\psi_{11}, \psi_{21}, \psi_{31}, \psi_{41}, \psi_{22}, \psi_{32}, \psi_{42}, \psi_{33}, \psi_{43}, \psi_{44}$) ordered {x, y, z, xy}. Using this input, the program makes the elastic loss tangent matrix $[\psi]$ symmetric. The input of $[\psi]$ and $[c]$ matrices should be such that the resulting imaginary part of $[c^*]$ is positive semidefinite.

Define the elastic loss tangent matrix via the **TB** family of commands, as follows:

1. Initialize the constant table (**TB,ELST**).

- Specify up to 21 constants (**TB**DATA).

3-D Input

Constant	Meaning
C1-C6	Terms $\psi_{11}, \psi_{21}, \psi_{31}, \psi_{41}, \psi_{51}, \psi_{61}$
C7-C12	Terms $\psi_{22}, \psi_{32}, \psi_{42}, \psi_{52}, \psi_{62}, \psi_{33}$
C13-C18	Terms $\psi_{43}, \psi_{53}, \psi_{63}, \psi_{44}, \psi_{54}, \psi_{64}$
C19-C21	Terms $\psi_{55}, \psi_{65}, \psi_{66}$

2-D Input

Constant	Meaning
C1-C4	Terms $\psi_{11}, \psi_{21}, \psi_{31}, \psi_{41}$
C7-C9	Terms $\psi_{22}, \psi_{32}, \psi_{42}$
C12-C13	Terms ψ_{33}, ψ_{43}
C16	Term ψ_{44}

5.3.7. Anisotropic Dielectric Loss Tangent

Current-technology coupled-field elements with piezoelectric capability use the **TB**,DLST table to input the matrix of dielectric loss tangents (factors). Anisotropic dielectric loss tangent is used in a harmonic piezoelectric analysis to account for anisotropic dielectric losses. For a list of elements that use this model, see [Material Model Element Support \(p. 5\)](#). For constitutive relations involving anisotropic electric loss tangent, see [Piezoelectrics in the Theory Reference](#). For more information on using **TB**,DLST in a piezoelectric analysis, see [Piezoelectric Analysis in the Coupled-Field Analysis Guide](#). Isotropic electric loss tangent can be input using **MP**,LSST.

The full 3 x 3 dielectric loss tangent matrix:

$$[\varphi] = \begin{bmatrix} \varphi_{11} & \varphi_{12} & \varphi_{13} \\ & \varphi_{22} & \varphi_{23} \\ & & \varphi_{33} \end{bmatrix}$$

is used together with the 3 x 3 dielectric permittivity matrix $[\epsilon]$ to form the imaginary part of the complex dielectric permittivity matrix $[\epsilon^*] = [\epsilon] - j[\phi] \cdot [\epsilon]$ in a harmonic piezoelectric analysis. The dielectric permittivity matrix $[\epsilon]$ can be defined by either specifying the permittivity constants (PERX, PERY, and PERZ) with **MP** commands or by specifying the terms of the permittivity matrix with the **TB**,DPER data table. The same dielectric loss tangent matrix $[\phi]$ applies to the permittivity at constant strain $[\epsilon_S]$ (input as **MP**,PERX/Y/Z or **TB**,DPER with *TBOPT* = 0) and to the permittivity at constant stress $[\epsilon_T]$ (input as **TB**,DPER with *TBOPT* = 1).

The order of rows and columns in matrix $[\phi]$ corresponds to the electric flux or electric field vector components ordered {x, y, z}. Input up to 6 constants ($\phi_{11}, \phi_{22}, \phi_{33}, \phi_{12}, \phi_{23}, \phi_{13}$) using the **TB**DATA command as shown below. For 2-D problems, a 2 x 2 matrix input needs up to 3 constants ($\phi_{11}, \phi_{22}, \phi_{12}$). Using this input, the program makes the dielectric loss tangent matrix $[\psi]$ symmetric.

Define the dielectric loss tangent matrix via the **TB** family of commands, as follows:

- Initialize the constant table (**TB**,DLST).

2. Specify up to 21 constants (**TB**DATA).

3-D Input

Constant	Meaning
C1-C6	$\phi_{11}, \phi_{22}, \phi_{33}, \phi_{12}, \phi_{23}, \phi_{13}$

2-D Input

Constant	Meaning
C1, C2, C4	$\phi_{11}, \phi_{22}, \phi_{12}$

5.4. Migration Model

The migration model (**TB**,MIGR) is used for modeling hydrostatic stress migration, thermomigration, and electromigration effects in diffusion analyses coupled with structural, thermal, and electric degrees of freedom, respectively.

Using these combined effects, **TB**,MIGR can be used for modeling electromigration of atoms (*TBOPT* = 0) or vacancies (*TBOPT* = 1) in metallic interconnects.

The following migration model topics are available:

5.4.1. Diffusion Flux and Chemical Potential

5.4.2. Atomic Flux Option (*TBOPT* = 0)

5.4.3. Vacancy Flux Option (*TBOPT* = 1)

See also [Electric-Diffusion Coupling](#), [Thermal-Diffusion Coupling](#), and [Structural-Diffusion Coupling](#) in the *Mechanical APDL Theory Reference*.

5.4.1. Diffusion Flux and Chemical Potential

The modified first Fick's law for the diffusion flux density J can be written in terms of chemical potential μ as follows:

$$J = -\frac{[D]}{kT} C \nabla \mu \quad (5.3)$$

where:

[D] = diffusivity matrix

C = concentration

T = absolute temperature

k = Boltzmann constant

For diffusion alone, the chemical potential can be written as $\mu = \mu_0 + kT \ln C$, and its substitution in [Equation 5.3 \(p. 251\)](#) produces the first Fick's law in the form of [Equation 9.1](#) (see [Diffusion in the Theory Reference](#)). When, in addition to diffusion, stress, temperature, and electric potential contribute to the process of migration, the expression for the generalized chemical potential of a particle can be written as:

$$\mu = \mu_0 + kT \ln C - \Omega \sigma_H + \frac{Q}{T} T + ZeV \quad (5.4)$$

where:

Ω = particle volume

σ_H = hydrostatic stress = $(\sigma_1 + \sigma_2 + \sigma_3) / 3$

Q = heat of particle transport

V = electric potential

Z = effective particle charge number

e = elementary charge

Substitution of Equation 5.4 (p. 252) in Equation 5.3 (p. 251) produces a flux J that includes both diffusion and migration of particles resulting from the hydrostatic stress gradient, temperature gradient, and electric field.

Using **TB,MIGR**, you can specify the option to model the flux of atoms (ions) (**TBOPT** = 0) or the flux of vacancies (**TBOPT** = 1).

5.4.2. Atomic Flux Option (**TBOPT** = 0)

The diffusion flux density J of atoms (ions) is coupled to the hydrostatic stress, temperature, and electric potential as follows:

$$J = -[D] \nabla C + \frac{[D]C\Omega}{kT} \nabla \sigma_H - \frac{[D]CQ}{kT^2} \nabla T - \frac{[D]CZe}{kT} \nabla V \quad (5.5)$$

where:

[D] = atomic diffusivity matrix

C = atomic concentration; normalized atomic concentration $\bar{C}$ if $C_{sat} > 1$ and $C = C_{sat} \bar{C}$

C_{sat} = saturated concentration (input as **MP,CSAT**)

Ω = atomic volume

An atom migrates in the direction of the hydrostatic stress gradient

Q = atomic heat of transport

Z = effective (or apparent) atom charge number

For electromigration applications, Z combines the electrostatic charge number (Z^e) and the electron wind charge number (Z^w). Z^e represents the electrostatic force acting on the atom (ion) in the direction of the electric field. Z^w represents the force resulting from the momentum exchange of the atom (ion) with the electrons drifting in the opposite direction of the electric field. The electron wind force is generally dominant, and Z is negative for atomic electromigration.

Alternatively:

$$J = -[D] \nabla C + \frac{[D]C\Omega}{RT} \nabla \sigma_H - \frac{[D]CQ}{RT^2} \nabla T - \frac{[D]CZF}{RT} \nabla V \quad (5.6)$$

where:

C = molar concentration

Ω = molar volume

Q = molar heat of transport

Z = effective atomic charge number

F = Faraday constant

R = universal gas constant

In addition to migration effects, **TB,MIGR** with the atom flux option (*TBOPT* = 0) can be used for modeling the effects of temperature and hydrostatic stress on the coefficients of diffusivity:

$$[D] = [D_0] e^{(-E_a + h\Omega\sigma_H)/kT} \quad \text{or} \quad [D] = [D_0] e^{(-E_a + h\Omega\sigma_H)/RT} \quad (5.7)$$

where:

$[D_0]$ = matrix of diffusivity coefficients (input on **MP**, DXX/DYY/DZZ)

E_a = activation energy of diffusion

h = atomic (or molar) volume multiplier for the calculation of hydrostatic stress effects on the diffusivity. Typically set to 1 for electromigration applications.

e = natural logarithm base

For the atomic flux option (*TBOPT* = 0), input the material data (**TBDATA**) as shown in the table below. The k and R notations in the table are defined as:

k = Boltzmann constant

R = universal gas constant

Constant	Meaning	Description	Equation
C1	E_a/k or E_a/R	E_a = activation energy	$[D] = [D_0] e^{(-E_a)/kT}$ or $[D] = [D_0] e^{(-E_a)/RT}$
C2	Ω/k or Ω/R	Ω = volume	$J = \frac{[D]C\Omega}{kT} \nabla \sigma_H$ or $J = \frac{[D]C\Omega}{RT} \nabla \sigma_H$
C3	Q/k or Q/R	Q = heat of transport	$J = -\frac{[D]CQ}{kT^2} \nabla T$ or $J = -\frac{[D]CQ}{RT^2} \nabla T$

Constant	Meaning	Description	Equation
C4	Ze/k or ZF/R	Z = effective charge number e = elementary charge F = Faraday constant	$J = -\frac{[D]CZ_e}{kT} \nabla V$ or $J = -\frac{[D]CZF}{RT} \nabla V$
C5	Not used	--	--
C6	Not used	--	--
C7	h	Volume multiplier for diffusivity dependence on hydrostatic stress	$[D] = [D_0] e^{h\Omega\sigma_H/kT}$ or $[D] = [D_0] e^{h\Omega\sigma_H/RT}$

Example 5.2: Defining an Atomic Migration Model

```

k=1.38e-23      ! Boltzmann's constant, J/K
k_eV=8.62e-5   ! Boltzmann's constant, eV/K

Va=2.71e-29    ! atomic volume, m^3
Ea=0.8         ! activation energy, eV
Qt=0.0094      ! heat of transport, eV
Ze=-23         ! effective charge number
h=1            ! hydrostatic stress multiplier for D

tb,migr,1,,,0  ! migration model with atomic flux option
tbdata,1,Ea/k_eV
tbdata,2,Va/k
tbdata,3,Qt/k_eV
tbdata,4,Ze/k_eV
tbdata,7,h
    
```

5.4.3. Vacancy Flux Option (TB0PT = 1)

The diffusion flux density J of vacancies has the following form:

$$J = -[D] \nabla C - \frac{[D]Cf\Omega}{kT} \nabla \sigma_H - \frac{[D]CQ}{kT^2} \nabla T - \frac{[D]CZe}{kT} \nabla V \quad (5.8)$$

where:

$[D]$ = vacancy diffusivity matrix

C = vacancy concentration; normalized vacancy concentration $\bar{C}$ if $C_{sat} > 1$ and $C = C_{sat} \bar{C}$

Ω = atomic volume

f = vacancy volume relaxation factor.

A vacancy has a volume $f\Omega$ and migrates in the direction opposite to the hydrostatic stress gradient. [\[Kirchheim\] \(p. 257\)](#)

Q = vacancy heat of transport

Z = effective (or apparent) charge number of the vacancy.

A vacancy migrates in the direction opposite to the electron wind; therefore, Z is positive for vacancy electromigration.

TB,MIGR with the vacancy flux option (*TBOPT* = 1) can also be used to specify the vacancy generation-annihilation rate G:

$$G = -\frac{C - C_e}{\tau_s} \quad (5.9)$$

where:

C = vacancy concentration; normalized vacancy concentration $\bar{C}$ if $C_{sat} > 1$ and $C = C_{sat} \bar{C}$

τ_s = characteristic time of vacancy generation-annihilation

C_e = equilibrium concentration of vacancies; normalized equilibrium concentration if $C_{sat} > 1$ and

$C_e = C_{sat} \bar{C}_e$

$$C_e = C_{0e} e^{h\Omega\sigma_H/kT} \quad (5.10)$$

where:

C_{0e} = equilibrium concentration in the absence of stress; normalized vacancy concentration if $C_{sat} > 1$ and $C_{0e} = C_{sat} \bar{C}_{0e}$

h = vacancy volume multiplier for the calculation of hydrostatic stress effects on the equilibrium concentration. Typically set to 1-f for vacancy electromigration.

e = natural logarithm base

For the vacancy flux option (*TBOPT* = 1), input the material data (**TBDATA**) as shown in the table below. The k and R notations in the table are defined as:

k = Boltzmann constant

R = universal gas constant

Constant	Meaning	Description	Equation
C1	E_a/k or E_a/R	E_a = activation energy	$[D] = [D_0] e^{(-E_a)/kT}$ or $[D] = [D_0] e^{(-E_a)/RT}$
C2	Ω/k or Ω/R	Ω = volume	$J = -\frac{[D]Cf\Omega}{kT} \nabla \sigma_H$ or $J = -\frac{[D]Cf\Omega}{RT} \nabla \sigma_H$
C3	Q/k or	Q = heat of transport	$J = -\frac{[D]CQ}{kT^2} \nabla T$ or

Constant	Meaning	Description	Equation
	Q/R		$J = -\frac{[D]CQ}{RT^2} \nabla T$
C4	Ze/k or ZF/R	Z = effective charge number e = elementary charge F = Faraday constant	$J = -\frac{[D]CZe}{kT} \nabla V$ or $J = -\frac{[D]CZF}{RT} \nabla V$
C5	Not Used	--	--
C6	Not Used	--	--
C7	h	Volume multiplier for the equilibrium concentration dependence on hydrostatic stress	$C_e = C_{0e} e^{h\Omega\sigma_H/kT}$
C8	f	Vacancy volume relaxation factor (defaults to 1)	$J = -\frac{[D]Cf\Omega}{kT} \nabla \sigma_H$ or $J = -\frac{[D]Cf\Omega}{RT} \nabla \sigma_H$
C9	C _{0e}	Equilibrium concentration at zero stress	$C_e = C_{0e} e^{h\Omega\sigma_H/kT}$
C10	τ_s	Characteristic vacancy generation or annihilation time	$G = -\frac{C - C_e}{\tau_s}$

Example 5.3: Defining a Vacancy Migration Model

```

k=1.38e-23      ! Boltzmann's constant, J/K
k_eV=8.62e-5    ! Boltzmann's constant, eV/K

Va=1.68e-29     ! atomic volume, m^3
Ze=0.84         ! effective vacancy charge number
f=0.6           ! vacancy volume relaxation factor
h=1-f           ! hydrostatic stress multiplier for Ce
Ce=1            ! normalized vacancy equilibrium concentration
ts=1.8e-3       ! characteristic time, s

tb,migr,1,,1    ! vacancy flux option
tbdata,2,Va/k
tbdata,4,Ze/k_eV
tbdata,7,h
tbdata,8,f
tbdata,9,Ce
tbdata,10,ts
    
```

Resources

The following resource offers more information about the vacancy flux model:

1. Kirchheim, R. "Stress and Electromigration in A1-Lines of Integrated Circuits." *Acta metall. mater.* Vol. 40, No. 2, pp. 309-323, 1992.

5.5. Thermal Properties

Thermal material properties are defined via the **TB, THERM** command. These include thermal conductivity (**TBOPT = COND**) and specific heat (**TBOPT = SPHT**). For **porous media** (p. 174), the fluid specific heat can also be defined (**TBOPT = FLSPHT**).

5.5.1. Thermal Conductivity (**TBOPT = COND**)

The thermal conductivity can be isotropic, orthotropic, or anisotropic. The full 3 x 3 thermal conductivity matrix is:

$$K = \begin{bmatrix} K_{xx} & K_{xy} & K_{xz} \\ K_{yx} & K_{yy} & K_{yz} \\ K_{zx} & K_{zy} & K_{zz} \end{bmatrix}$$

Define up to nine constants:

Isotropic thermal conductivity matrix -- Define K_{xx} only. (The K_{yy} and K_{zz} values are assumed to be K_{xx} .)

Orthotropic thermal conductivity matrix -- Enter three constants: K_{xx} , K_{yy} , and K_{zz} . The program assigns values of zero to the off-diagonal conductivity constants (K_{ij} , $i \neq j$).

Anisotropic thermal conductivity matrix -- If you define the first six constants ($K_{xx}, K_{yy}, K_{zz}, K_{xy}, K_{xz}, K_{yz}$), the program makes the conductivity matrix symmetric (that is, $K_{xy} = K_{yx}$, $K_{xz} = K_{zx}$, $K_{yz} = K_{zy}$).

General nonsymmetric anisotropic thermal conductivity matrix -- Define all nine constants.

Constant	Meaning
C1	K_{xx}
C2	K_{yy}
C3	K_{zz}
C4	K_{xy}
C5	K_{xz}
C6	K_{yz}
C7	K_{yx}
C8	K_{zx}
C9	K_{zy}

All defined constants are based on the element coordinate system. By default, the element coordinate system is the global coordinate system.

For thermal elements, only the first three constants (C1, C2, and C3) can be used to define isotropic or orthotropic thermal conductivity.

For coupled pore-pressure-thermal elements, all nine constants (C1 to C9) can be defined.

Define temperature- or field-dependent data for the data table via the **TBTEMP** or **TBFIELD** commands, respectively.

5.5.2. Specific Heat (**TBOPT = SPHT**)

Define specific heat via **TB,THERM,,,,SPHT**. Input the specific-heat value as C1 on the **TBDATA** command.

For [porous media \(p. 174\)](#), this is considered to be the solid-skeleton specific heat.

5.5.3. Fluid Specific Heat (**TBOPT = FLSPHT**)

For [porous media \(p. 174\)](#), you can define fluid specific heat via **TB,THERM,,,,FLSPHT**. Input the fluid specific-heat value as C1 on the **TBDATA** command.

Chapter 6: Material Curve-Fitting

Material curve-fitting enables you to derive coefficients from experimental data that you provide for your material. Curve fitting involves comparing your experimental data to certain preexisting nonlinear material models. With this capability, you compare experimental data versus program-calculated data for different nonlinear models; based on those comparisons, you can determine the best material model to use during solution.

Curve-fitting is based on the material data table configurations (**TB**). Data manipulation and construction are performed separately (**TBFT**).

The following material curve-fitting topics are available:

- 6.1. Hyperelastic Material Curve-Fitting
- 6.2. Viscoelastic Material Curve-Fitting
- 6.3. Chaboche Material Curve-Fitting
- 6.4. Creep Material Curve-Fitting

6.1. Hyperelastic Material Curve-Fitting

Hyperelastic curve-fitting is a tool for estimating your material constants by inputting your experimental data and comparing it to the hyperelastic material models. Your stress-strain curves can be converted to any of the supported hyperelastic models, including Mooney-Rivlin, Ogden, Neo-Hookean, Polynomial, Arruda-Boyce, Gent, and Yeoh. Compressible hyperelastic Ogden hyper-foam and Blatz-Ko models are also supported.

To perform curve-fitting, you input experimental data, select a model from one of the provided hyperelastic models, perform a regression analysis, graphically view the curve-fitting results, compare the fits to the experimental data, and write the fitted coefficients to the database as nonlinear data table commands for the subsequent finite element analyses.

Hyperelastic models can define three types of behavior: purely incompressible, nearly incompressible, and compressible. Hyperelastic curve-fitting is based on the HYPER option (**TB**).

Curvefitting for fully- or nearly-incompressible hyperelastic material models use the incompressibility equation $\lambda_1 * \lambda_2 * \lambda_3 = 1$ (where λ is the stretch) to evaluate deviatoric material parameters. The volumetric coefficients are not used during the evaluation of the coefficients that define the deviatoric terms for these models. For more information, see [Hyperelasticity in the Theory Reference](#).

The following topics concerning hyperelastic material curve-fitting are available:

- 6.1.1. Understanding the Hyperelastic Material Curve-Fitting Process
- 6.1.2. Step 1. Prepare Hyperelastic Experimental Data
- 6.1.3. Step 2. Input the Hyperelastic Experimental Data
- 6.1.4. Step 3. Select a Hyperelastic Material Model Option
- 6.1.5. Step 4. Initialize the Hyperelastic Coefficients

6.1.6. Step 5. Specify Hyperelastic Control Parameters and Solve

6.1.7. Step 6. Plot Your Hyperelastic Experimental Data and Analyze

6.1.8. Step 7. Write Hyperelastic Curve-Fitting Data to the Database

6.1.1. Understanding the Hyperelastic Material Curve-Fitting Process

The steps for hyperelastic curve-fitting are defined as follows:

1	Step 1. Prepare Hyperelastic Experimental Data (p. 260)	The experimental data must exist in a plain text file delimited by a space or a comma.
2	Step 2. Input the Hyperelastic Experimental Data (p. 261)	Read in your experimental data by specifying the file name and path.
3	Step 3. Select a Hyperelastic Material Model Option (p. 262)	The material options for the applicable curve-fitting regimen are defined via TB . Nine hyperelastic models are supported. After you select a model, you can still change to another model if an ideal fit is not realized.
4	Step 4. Initialize the Hyperelastic Coefficients (p. 263)	Depending on the model, hyperelastic curve-fitting can be a linear or nonlinear regression process. The hyperelastic material models, along with the associated process for each are listed in Table 6.3: Hyperelastic Curve-Fitting Model Types (p. 262) .
5	Step 5. Specify Hyperelastic Control Parameters and Solve (p. 265)	Set the error norm to be used to generate the curve fit.
6	Step 6. Plot Your Hyperelastic Experimental Data and Analyze (p. 266)	Review and verify the results by comparing the experimental data and the regression errors. If the results are not acceptable, repeat steps 3 to 5 to perform a new curve-fitting solution.
7	Step 7. Write Hyperelastic Curve-Fitting Data to the Database (p. 266)	Write your curve-fitting results to the database in the TB material data table format.

6.1.2. Step 1. Prepare Hyperelastic Experimental Data

Curve fitting requires experimental test data. Your hyperelastic curve-fitting data must be a comma or space delimited file, referencing your stress vs. strain values.

Hyperelastic curve-fitting supports three main behaviors:

- Case 1 - Totally Incompressible Models (see [Table 6.1: Experimental Details for Case 1 and 2 Models and Blatz-Ko \(p. 261\)](#))
- Case 2 - Nearly Incompressible Models (see [Table 6.1: Experimental Details for Case 1 and 2 Models and Blatz-Ko \(p. 261\)](#))
- Case 3 - Compressible Models (see [Table 6.2: Experimental Details for Case 3 Models \(p. 261\)](#))

The types of data required for each of these cases is defined in the tables below.

Table 6.1: Experimental Details for Case 1 and 2 Models and Blatz-Ko

Experimental Type	Column 1	Column 2	Column 3
Uniaxial Test	Engineering Strain	Engineering Stress	
Biaxial Test	Engineering Strain	Engineering Stress	
Planar/Shear Test	Engineering Strain (in loading direction)	Engineering Stress	
Simple Shear Test	Engineering Shear Strain	Engineering Shear Stress	(Optional) Engineering Normal Stress (normal to the edge of shear stress)
Volumetric Test	Volume Ratio (J)	Hydrostatic Pressure	

Table 6.2: Experimental Details for Case 3 Models

Experiment Type	Column 1	Column 2	Column 3
Uniaxial Test	Engineering Strain	Lateral Direction Engineering Strain	Engineering Stress
Biaxial Test	Engineering Strain	Engineering Strain (in thickness direction)	Engineering Stress
Shear Test	Engineering Strain (in loading direction)	Engineering Strain (in thickness direction)	Engineering Stress
Simple Shear Test	Engineering Shear Strain	Engineering Shear Stress	(Optional) Engineering Normal Stress (normal to the edge of shear stress)
Volumetric Test	Volume Ratio (J)	Hydrostatic Pressure	

J is the ratio of current volume to the original volume.

All stresses output in POST1/POST26 are true stresses and logarithmic strains.

6.1.3. Step 2. Input the Hyperelastic Experimental Data

Use the EADD option (**TBFT**) to input your data files. First, designate whether they are uniaxial, biaxial, shear, simple shear, or volumetric, and then designate the location in the *Option2*, *Option3*, and *Option4* fields. All of the stress values are engineering stress except for the volumetric option (true stress). The file should be a simple, delimited set of stress and strain values similar to the following:

```
0.9703 60.00
0.9412 118.2
0.9127 175.2
0.8847 231.1
```

For temperature-dependent curve-fitting, specify your temperature values at the top of the experimental data using the

```
/temp,value
```

line. This header format specifies the attribute (temp) and its value (100). An example of a typical data input using these attributes follows:

```
/temp,100
0.9703 60.00
0.9412 118.2
0.9127 175.2
0.8847 231.1
```

Adding this header introduces a temperature attribute of 100 degrees. You can add additional data sets at other temperatures, in additional files. One file can have data at only one temperature.

For compressible materials, the default curve-fitting tool behavior is to solve only for stress as a function of strain and lateral strain. To force the curve-fitting tool to also fit experimental lateral strain data to generate the coefficients for the [Ogden compressible foam](#) model, add the line **/USEL, 1** near the top of the experimental data file. This option is valid for uniaxial, biaxial and shear test data.

6.1.3.1. Syntax and Arguments

```
TBFT,EADD,ID,Option1,Option2,Option3,Option4 ! experimental data input
```

where:

ID = Index corresponding to the material number
Option1 = UNIA, BIAx, SHEA, SSHE or VOLU
Option2 = name of file containing experimental data
Option3 = file name extension
Option4 = file directory

6.1.4. Step 3. Select a Hyperelastic Material Model Option

Use the FADD option (**TBFT**) to define your constitutive model. [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) below lists the models that are available for hyperelastic curve-fitting. When volumetric data is provided, a compressible or nearly incompressible model is implied, and only the designated options are available. When no volumetric data is provided, the model is understood to be incompressible, and those model options will be available. Providing zero coefficients for the volumetric data field also denote an incompressible model.

Table 6.3: Hyperelastic Curve-Fitting Model Types

Model Name	Abbreviation	Order/Options	No. of Coefficients [1]	Linear/Nonlinear Fitting
Mooney-Rivlin	moon	2, 3, 5, 9	2/3/5/9+1	Linear
Polynomial	poly	1 to N	see below [2]	Linear
Yeoh	yeoh	1 to N	N+N	Linear
Neo-Hookean	neoh	-	1+1	Linear
Ogden	ogde	1 to N	2*N+N	Nonlinear
Arruda-Boyce	boyc	-	2+1	Nonlinear
Gent	gent	-	2+1	Nonlinear

Model Name	Abbreviation	Order/Options	No. of Coefficients [1]	Linear/Nonlinear Fitting
Blatz-Ko	blat	-	1	Nonlinear
Ogden Hyper-foam	foam	1 to N	2*N+N	Nonlinear

1. The number of coefficients is usually the sum of the number of deviatoric coefficients and the number of volumetric coefficients.
2. The number of coefficients for a polynomial depend on the polynomial order N.

$$\text{Number of Coefficients} = \sum_{i=1}^N (1+i) + N$$

Blatz-Ko and Ogden hyper-foam are compressible models. For Ogden hyper-foam, the experimental data you provide requires additional fields.

For more information about the hyperelastic models available for curve-fitting, see [Hyperelasticity \(p. 83\)](#) in the [Material Reference \(p. 1\)](#).

6.1.4.1. Syntax and Arguments

```
TBFT, FADD, ID, HYPER, Option2, Option3 ! material model options input
```

where:

ID = Index corresponding to the material number

Option2 = model name, as specified in [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above)

Option3 = order or number of coefficients, where applicable. [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) specifies the number and type of coefficient(s) necessary for each hyperelastic model type.

6.1.5. Step 4. Initialize the Hyperelastic Coefficients

Depending on the model you choose, hyperelastic curve-fitting can be a linear or a nonlinear regression process. In both cases, the initial coefficients that you provide determine the accuracy and efficiency of the curve fit. The initial values of the coefficients generally come from experience, and also from examining the function that defines the model to which you are attempting to compare/fit your data. For most hyperelastic models, 1 or -1 is a good starting point; however, coefficient values can vary greatly depending on the model chosen. The Gent model, for example, provides good fit with initial coefficient values as high as 1000.

You can also fix (hold constant) your coefficients (**TBFT** with *Option4* = FIX). You specify a value for a coefficient and keep it unchanged, while allowing the other coefficients to be operated on. You can then release the fixed coefficient and operate on it after the others are optimized. By default, all coefficients are free to vary. The capability to fix coefficients applies only to nonlinear curve fitting (as listed in [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#)).

For temperature-dependent experimental data, enable temperature-dependency and specify a reference temperature before solving for the coefficients. You can set the reference temperature only to values specified using the */temp,value* header line in the experimental data.

You can also initiate multiple solves to evaluate coefficients at all available discrete temperature values (**TBFT** with TREF = ALL). In this case, for data at three temperatures (t1, t2, and t3), a single **TBFT,SOLVE** command performs three separate solve operations at those three discrete temperatures.

With temperature dependency on and the reference temperature set to a particular value, a **TBFT,SOLVE** command solves for coefficients only at that temperature. To solve for coefficients at other temperatures, you set the reference temperature to each desired discrete temperature value and solve again.

You can initialize the coefficients before or after enabling temperature dependency. If the coefficients are initialized before enabling temperature dependency, the specified coefficients become the initial coefficients for all future solves for that particular model. These coefficients are, however, overridden when temperature dependency is enabled and another set of initial values are specified at discrete temperature values. The curve-fitting tool searches for the initial coefficients at a given temperature; if no coefficients are specified at a discrete temperature, it uses the initial coefficients set before temperature dependency was enabled.

6.1.5.1. Syntax and Arguments

```
TBFT,SET,ID,HYPER,Option2,Option3,Option4,Option5 ! initialize coefficients
```

where:

ID = Index corresponding to the material number

Option2 = model name (see [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for the available models)

Option3 = order, if applicable

Option4 = index of coefficient

Option5 = value of the coefficient

To set *Option4* for a reference temperature, or for temperature dependency:

```
TBFT,SET,ID,HYPER,Option2,Option3,Option4,Option5
```

where:

ID = Index corresponding to the material number

Option2 = model name. See [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for the available models.

Option3 = order, if applicable

Option4 = TDEP or TREF

Option5 = If *Option4* = TDEP, *Option5* = 1 enables temperature dependency, and *Option5* = 0 disables it. If *Option4* = TREF, *Option5* is either ALL or the reference temperature.

```
TBFT,FIX,ID,HYPER,Option2,Option3,Option4,Option5
```

where:

ID = Index corresponding to the material number

Option2 = model name See [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for the available models.

Option3 = (Blank--not applicable)

Option4 = index of coefficient

Option5 = 1 for fixed and 0 to vary

6.1.6. Step 5. Specify Hyperelastic Control Parameters and Solve

Depending on the model, your hyperelastic curve-fitting is either a linear or nonlinear regression process (see [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for details). Your error norms can be either normalized or non-normalized. Normalized error norms (the default regression option) generally give better results than the non-normalized error norms, since normalized error gives equal weight to all of your data points.

The solution control parameters of a nonlinear regression include:

1. Number of iterations
2. Residual tolerance
3. Coefficient change tolerance

The solution stops when both the residual tolerance and the coefficient change tolerance of your error norm are met, or if the number of iterations criteria is met. When you use nonlinear regression, you must initialize your coefficients.

6.1.6.1. Syntax and Arguments

```
TBFT, SOLVE, ID, HYPER, Option2, Option3, Option4, ..., Option7 ! set control parameters and solve
```

where:

ID = Index corresponding to the material number

Option2 = model name. See [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for the models available.

Option3 = order or number of your coefficients. See [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for possible values.

Option4 = curve-fitting procedure: 0 = non-normalized least squares, 1 = normalized least squares

Option5 = maximum number of iterations

Option6 = tolerance of residual changes

Option7 = tolerance of coefficient changes

Other solution parameters are available (**TBFT**).

6.1.7. Step 6. Plot Your Hyperelastic Experimental Data and Analyze

After solving, the coefficient tables contain the fitted coefficients and also the residual errors. You can plot your data and visually interpret the results. The first column is always the X- axis, with each additional column plotted separately as a function of column one.

Try to reserve column one for the variable that you would like to see vary in the plot. For example, if your data contains time, temperature, stress, and creep strain, and you want to see creep strain vary as a function of time at different temperatures and stresses in the plot, add your experimental data via multiple **TBFT**,EADD commands (by splitting the file into multiple experimental files as prescribed earlier, one for each combination of temperature and stress).

6.1.7.1. Plotting

Use the **GRAPH** button to plot the data. Your plots show column 2 and above as separate curves, plotted as a function of column 1. The data in column 1 is always the X-axis. By default, all the experiments are plotted in the GUI window. To view specific data and its corresponding fitting result, you can click the right mouse button (RMB) on the specific data set, and pick a desired option to view the results. Other RMB plotting utilities can be found for different data fields in the curve-fitting GUI window.

Use RMB functions to Zoom, Fit, Save Plot to File, View/Hide Legend and View/Hide Grid. Two or more fitted functions can also be compared in the same plot. For example, you can view Mooney2 Uniaxial and Mooney9 Uniaxial plots directly on top of each other, provided you have already solved for both of these functions. RMB also allows you to set the number of points used to generate the plot, and also change the minimum X value and the maximum X value in a plot. You can also hide a particular curve within a graph, turn the legend and/or axis displays on and off, and switch the scales between log scale and regular scale. Use the middle-mouse button to eliminate a specific curve and clarify or refine the remaining curve.

6.1.7.2. Review/Verify

The two factors you consider in determining results acceptability are visual fit and the error norm/residual values. When you plot the curve, the error norm/residual values are printed in the curve-fitting GUI window. Error norm values help you determine the quality of curve-fitting and whether to accept the results, but are not always the best indicator of a valid curve fit. Plotting the curves and visually assessing the result is usually the best indication. If the results are unacceptable, you may want to go back to [step 3 \(p. 262\)](#) and solve again, either by picking a different model, increasing the order, or redefining your initial values of the coefficients or other control parameters. You can continue to use your original experimental data, repeating [step 3 \(p. 262\)](#) through [step 7 \(p. 266\)](#) until you get an acceptable solution.

6.1.8. Step 7. Write Hyperelastic Curve-Fitting Data to the Database

When you are satisfied with the curve-fitting results, write the curve-fitting data to the database (**TBFT**,FSET). The program converts the coefficients to the appropriate **TB** table format before writing to the database, storing the data as part of the material property set for use in subsequent analyses.

6.1.8.1. Syntax and Arguments

```
TBFT, FSET, ID, HYPER, Option2, Option3 ! write data to TB
```

where:

ID = Index corresponding to the material number

Option2 = model name. See [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for the models available.

Option3 = order or number of your coefficients. See [Table 6.3: Hyperelastic Curve-Fitting Model Types \(p. 262\)](#) (above) for possible values.

6.2. Viscoelastic Material Curve-Fitting

Viscoelastic material curve-fitting determines your material constants by relating your experimental data to the Prony series expansion for both the shear and bulk modulus of the hypoviscoelastic material option.

To perform curve-fitting, you input the experimental data, define the order of Prony series expansion, perform nonlinear regression, view the curve-fitting results graphically, compare the experimental data, and write the fitted coefficients to the database as nonlinear data table commands for the subsequent finite elements analyses. You can fit shear modulus and/or bulk modulus and/or shift functions, along with discrete temperature dependencies for multiple data sets.

The following topics concerning viscoelastic material curve-fitting are available:

[6.2.1. Understanding the Viscoelastic Material Curve-Fitting Process](#)

[6.2.2. Step 1. Prepare Viscoelastic Experimental Data](#)

[6.2.3. Step 2. Input the Viscoelastic Data](#)

[6.2.4. Step 3. Select a Viscoelastic Material Model Option](#)

[6.2.5. Step 4. Initialize the Viscoelastic Coefficients](#)

[6.2.6. Step 5. Specify Viscoelastic Control Parameters and Solve](#)

[6.2.7. Step 6. Plot the Viscoelastic Experimental Data and Analyze](#)

[6.2.8. Step 7. Write Viscoelastic Curve-Fitting Data to the Database](#)

6.2.1. Understanding the Viscoelastic Material Curve-Fitting Process

The steps for viscoelastic curve-fitting are defined as follows:

1	Step 1. Prepare Viscoelastic Experimental Data (p. 268)	The experimental data must be a plain text file delimited by a space or a comma.
2	Step 2. Input the Viscoelastic Data (p. 269)	Read in the experimental data as a plain text file.
3	Step 3. Select a Viscoelastic Material Model Option (p. 270)	This includes Prony series expansion of shear and/or bulk moduli as well as shift function. The supported shift functions include WLF and TN.
4	Step 4. Initialize the Viscoelastic Coefficients (p. 270)	Viscoelastic curve-fitting is a nonlinear regression; the initial value of your coefficients is very important for a successful solution.
5	Step 5. Specify Viscoelastic Control Parameters and Solve (p. 273)	Specify the error norm to be used, the solution control parameters, and perform the nonlinear regression.

6	Step 6. Plot the Viscoelastic Experimental Data and Analyze (p. 276)	Graphically view the curve-fitting results. Review and verify the results by comparing them with the experimental data and the regression errors. If any factor is not acceptable, repeat steps 3 to 7 to obtain a new curve-fitting solution.
7	Step 7. Write Viscoelastic Curve-Fitting Data to the Database (p. 276)	Write your curve-fitting results in TB command format to the database.

6.2.2. Step 1. Prepare Viscoelastic Experimental Data

Viscoelastic curve-fitting allows for experimental data in both the time and frequency domains.

To input experimental data for viscoelastic materials in the time domain, the data must be shear modulus and/or bulk modulus as a function of time and temperature. The experimental data is named **sdec** or **bdec** to distinguish it from other data types, such as hyperelastic or creep, and to allow convenient analysis documentation.

To input experimental data for viscoelastic materials in the frequency domain, the data must be storage modulus and loss modulus as a function of frequency, and optionally temperature. The experimental data is named **sdec** or **bdec**. The data must be represented in three columns, and a **/1, freq** entry at the top of the experimental data file is necessary to indicate that column 1 is frequency. Similar to time domain data, it is possible to input data for shear (experiment type **sdec**) and bulk behavior (experiment type **bdec**). Frequency values are entered in cycles-per-second (which the program converts to radians-per-unit-time).

Your viscoelastic test data must be a plain text file with headers to define the test data. The data file should be in table form, delimited by a space or a comma. The header defines the test data type and the temperature for your test data. For viscoelastic curve-fitting with multiple temperatures, you can evaluate coefficients at each discrete temperature point and write it as a temperature-dependent Prony data table, or you can use the Williams-Landau-Ferry (WLF) or Tool-Narayanaswamy (TN) shift functions to account for the temperature dependency. (See [Shift Functions](#) in the *Mechanical APDL Theory Reference*.) A separate data file must be provided for each discrete temperature. The viscoelastic test data can be any of 4 data types, see [Table 6.4: Viscoelastic Data Types and Abbreviations \(p. 268\)](#).

Table 6.4: Viscoelastic Data Types and Abbreviations

Time	time
Shear modulus	smod
Bulk modulus	bmod
Temperature	temp

The headers are used to describe the data types that characterize the test data columns or attributes of the data.

The following listing contains the appropriate headers, followed by the delimited data:

```
/temp,100 ! define temperature attribute
0.01 2992.53
1 2978.514207
2 2965.45541
4 2942.293214
6 2922.530649
8 2905.612202
10 2891.073456
```

```

20 2842.506984
40 2798.142793
60 2772.383729
80 2750.631843
100 2730.398114
200 2643.125432
400 2517.475394
600 2431.262053
800 2366.580897
1000 2313.955396
2000 2117.922594
4000 1833.734397
6000 1627.199197
8000 1470.6806
10000 1347.264527
20000 964.0141125
40000 586.1405449
60000 392.186777
80000 277.2706253
100000 202.0025278
200000 46.87056342
400000 2.669209118
600000 0.156653269
800000 0.0137224
1000000 0.005591539

```

6.2.3. Step 2. Input the Viscoelastic Data

Use the EADD option (**TBFT**) to input your data files.

The experimental data must be read in from a plain text file. The data must be [prepared properly \(p. 268\)](#) and include both the header information and the formatted test data. Each file represents a data set, and can be the complete data set or part of a larger data set.

You can include several data sets, including tests performed at different temperatures. Although different data sets can have the same/or different temperature, each file can have only one temperature. Multiple temperature data sets must be input with multiple files.

Two types of data may be required for viscoelastic curve-fitting, either shear modulus vs. time and/or bulk modulus vs. time. The data can also be a function of temperature, which can then be accounted for by either WLF or TN shift function.

6.2.3.1. Syntax and Arguments

```
TBFT, EADD, ID, Option1, Option2, Option3, Option4 ! input data
```

where:

ID = index corresponding to the material number
Option1 = experimental data type, either sdec or bdec
Option2 = name of file containing experimental data
Option3 = file name extension
Option4 = file directory

The sdec coefficient refers to the shear modulus as a function of time. The bdec coefficient refers to the bulk modulus as a function of time.

6.2.4. Step 3. Select a Viscoelastic Material Model Option

The **TBFT** command provides the curve-fitting tool for viscoelastic material modeling. The viscoelastic material behavior is represented by a set of Prony-series expansions of shear and/or bulk moduli to characterize the shear and the bulk deformation of the material. Shift functions are available to characterize the material's temperature dependency.

Define a case name to associate the set of coefficients for the Prony expansions with the shift functions that characterize the material behavior. You can use the case name to define several different options that characterize the same test data, and then to compare the curve-fitting results. To define the material model, define a case name, then specify the order of shear and bulk moduli and the type of the shift function(s), if required. You need to create an additional case to define different shear order, bulk order or shift options. After creating a case, the number of shear terms, bulk terms, or shift options cannot be changed.

6.2.4.1. Syntax and Arguments

You define a viscoelastic material with the Prony series expansion by creating a case (**TBFT**,FCASE) and setting the order of shear modulus, bulk modulus, and shift options. The first line includes FCASE,ID,NEW. Then you specify the number of shear terms, bulk terms, and the shift function. The case is actually created only after the option is issued. The following syntax examples and argument descriptions illustrate the relationship of these activities:

```
TBFT, FCASE, ID, NEW, Option2, Option3
```

where:

ID = Index corresponding to the material number
Option2 = PVHE (refers to Prony Viscohyoelastic)
Option3 = Case name

```
TBFT, FADD, ID, CATEGORY, Option2, Option3
```

where:

ID = Index corresponding to the material number
CATEGORY = VISCO
Option2 = pshear or pbulk or shift
Option3 = dependent on *Option2* as follows:

- *Option2* = pshear or bulk, *Option3* = NONE, or 1 to N
- *Option2* = shift, *Option3* = NONE or TN or WLF

```
TBFT, FCASE, ID, FINI
```

6.2.5. Step 4. Initialize the Viscoelastic Coefficients

The initial values chosen for your coefficients determine the success of your viscoelastic curve-fitting operations. A complete model has $(2 \cdot N_G + 1) + (2 \cdot N_k + 1) + N_S$ number of the coefficients, where:

- N_G is the order of the Prony series expansion of the shear modulus,
- N_K is the order of the Prony series expansion of the bulk modulus, and
- N_S is the number of coefficients of the shift function. ($N_S = 2$ for the TN option, and $N_S = 3$ for the WLF option.)

The coefficients are ordered as shear terms first, then the bulk terms, and then the shift function. The coefficients are ordered as $A_0^G, A_1^G, \tau_1^G, A_2^G, \tau_2^G, \dots, A_n^G$, and τ_n^G for shear modulus, and $A_0^K, A_1^K, \tau_1^K, A_2^K, \tau_2^K, \dots, A_n^K$, and τ_n^K for bulk modulus.

A shift function must be used together with your shear and/or bulk modulus for temperature-dependent experimental data.

The default coefficient is set to 1, but it is good practice to redefine the initial values before solving. When initializing your coefficients, set A_n^K s to 1 and τ_n^K s to time values that are equally distributed in the log scale, spanning the data range from minimum to maximum time for the time domain data.

For example, consider the shear-decay versus time-data file. If the time values vary from 1 to 10000, and if you use third-order Prony, logical guesses for τ_1^G, τ_2^G and τ_3^G that span this range could be $\tau_1^G = 1, \tau_2^G = 100$, and $\tau_3^G = 10,000$ (also (1), (10) and (10,000) or (1), (1,000) and (10,000), respectively).

For the frequency domain data, a trial-and-error process is necessary. Enter τ values at different orders of magnitude (1E-5, 1E-3, 1E-1, 1E1, 1E2. ...). The A (alpha values) values can be set to 1, and the least-squares solver generally converges to the best possible alpha values irrespective of the initial values.

The following equations describe the relationship between the Prony Coefficient (α_n) and the corresponding coefficient generated in curve-fitting (A_n). K and G are the shear modulus and bulk modulus at $t = 0$. This was done to keep all α_n^G and α_n^K values used in the **TB** tables positive.

$$\alpha_n^K = (A_n^K * A_n^K) / K$$

$$\alpha_n^G = (A_n^G * A_n^G) / G$$

A good guess for the WLF or TN parameter is the reference temperature you used during your partial solve for shear and bulk. The index of the reference/base temperatures is the sum of *NumShear* + *NumBulk* + 1.

You can also fix (hold constant) your coefficients. You specify a value for a coefficient and keep it unchanged, while allowing the other coefficients to be operated on. You can then release the fixed coefficient later if desired. By default, all of the coefficients are free to vary.

You estimate coefficients for temperature-dependent data either by using the [shift function \(p. 274\)](#) or by setting the temperature-dependency flag and setting a reference temperature before solving for the coefficients. You can set the reference temperature only to values specified using the /temp,value header line in the experimental data.

You can also specify TREF = ALL and initiate multiple solves to evaluate coefficients at all available discrete temperature values. In this case, for data at three temperatures (t_1, t_2 , and t_3), a single

TBFT,SOLVE command initiates three separate solve operations at those three discrete temperature values, and generate data at three corresponding discrete temperatures.

With temperature dependency specified and the reference temperature set to a particular value, a **TBFT,SOLVE** command solves for coefficients only at that discrete temperature. To solve for coefficients at other temperatures, set the reference temperature to each of the desired discrete temperature values and solve again.

You can initialize the coefficients before or after activating temperature dependency. If the coefficients initialize before setting temperature dependency, the specified coefficients become the initial coefficients for all future solves for that particular model. These coefficients are, however, overridden when temperature dependency is active and another set of values is specified at a discrete temperature value. The curve-fitting tool looks for the initial coefficients at a particular temperature. If no coefficients are specified at discrete temperature values, the initial coefficients set before temperature dependency was activated are used.

6.2.5.1. Syntax and Arguments

```
TBFT, SET, ID, CASE, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = Index of coefficient

Option5 = Value of coefficient

Example 6.1: Initializing Coefficients

```
TBFT, SET, 1, myvisco1, 1, 1.2 ! Initialize the first coefficient to 1.2
TBFT, SET, 1, myvisco1, 2, 1.5 ! Initialize the second coefficient to 1.5
```

By default, coefficients are not fixed. You can fix a coefficient (to a value set via **TBFT,SET**) or release a previously fixed coefficient (**TBFT,FIX**).

```
TBFT, FIX, ID, CASE, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = Index of coefficient

Option5 = 1 to fix, 0 to vary

Example 6.2: Fixing Coefficient Values

```
TBFT, FIX, 1, myvisco1, 1, 1. ! Fix the first coefficient to a value set via TBFT, SET
TBFT, FIX, 1, myvisco1, 2, 1 ! Fix the second coefficient to a value set via TBFT, SET
```

Temperature dependency uses *Option4*, and references data files you entered with the appropriate **temp** header.

```
TBFT, SET, ID, VISC0, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = Model name. See [Table 6.4: Viscoelastic Data Types and Abbreviations \(p. 268\)](#) (above) for the models available.

Option3 = (Blank--not applicable)

Option4 = TDEP or TREF

Option5 = If *Option4* = TDEP, then 1 enables temperature dependency and 0 disables it. If *Option4* = TREF, this value is a specific temperature, or ALL.

6.2.6. Step 5. Specify Viscoelastic Control Parameters and Solve

Viscoelastic curve-fitting is a nonlinear regression process. You can use either normalized or non-normalized error norm for the regression. The normalized error norm is the default regression option for calculating the error. This error norm generally gives better results than the non-normalized error norm, since the normalized error gives equal weight to all data points.

The solution control parameters for nonlinear regression include number of iterations, residual tolerance, and coefficient change tolerance. The solution stops when both residual tolerance of error norm and coefficient change tolerance is met or if the number of iterations criteria is met. The coefficients are updated when the solution is completed. In general it is very difficult to directly solve a complete case including coefficients of the shear modulus, the bulk modulus and the shift function. Three solver options including shear modulus only, bulk modulus, and shift function (or all) are provided to allow you to solve only Prony coefficients of the shear modulus, Prony coefficients of the bulk modulus, and coefficients of the shift function. In many cases, however, the coefficients of shift function can't be solved until the shear or bulk modulus are solved. It is normal for a solution to not converge at first, but to stop when the maximum iterations criteria is reached. At that point, you should examine the curve-fitting results and the solution history before proceeding any further. You can then adjust parameters and resolve the problem whenever it is necessary.

6.2.6.1. Viscoelastic Parameter Scaling

To facilitate convergence, you can activate parameter scaling (*Option9* = 1 on **TBFT,SOLVE**).

Parameters for Prony-series curve-fitting can have a wide range of values. Prony Alpha parameters can be of different orders, varying from initial shear or bulk modulus to a very small value. Optimization algorithms used by the curve-fitting tool perform better when all parameters are scaled to a similar range.

Parameter scaling replaces an Alpha parameter with EXP(Alpha) in the curve-fitting equations to improve the curve-fitting optimization process. The coefficients fit and estimated by the curve-fitting tool have different values during the curve-fitting process; however, the values are rescaled correctly when the parameter information is written in the appropriate **TB** table format to the database (**TBFT,FSET**).

6.2.6.2. Temperature-Dependent Solutions Using the Shift Function

For viscoelastic curve-fitting using the shift function, follow these three steps to perform the regression:

1. Solve the shear coefficients (if any):

Set the partial solve option (**TBFT**,SET,,,,COMP,PSHEAR).

Set the reference temperature at which your partial solution is performed (**TBFT**,SET,,,,TREF,TX). Only data at temperature TX is used to estimate shear coefficients

Solve

2. Solve the bulk coefficients (if any):

Set the partial-solve option (**TBFT**,SET,,,,COMP,PBULK).

Set the reference temperature at which your partial solution is performed (**TBFT**,SET,,,,TREF,TX). The reference temperature should be the same for both shear and bulk. Only data at temperature TX is used to estimate shear coefficients

Solve (**TBFT**,SOLVE).

3. Solve the shift function (or all) coefficients:

Set the partial solve option (**TBFT**,SET,,,,COMP,PVHE).

TREF is not used when solving for all parameters. All temperature data is used to estimate the coefficients.

TBFT,SOLVE

6.2.6.3. Temperature-Dependent Solutions Without the Shift Function

If you do not use the shift function, discrete data must be supplied to account for temperature dependency. Follow these steps to perform the regression:

1. Set the temperature-dependency flag (**TBFT**,SET,,,,TDEP,1).

2. Solve the shear coefficients (if there are any):

Set the partial-solve option (**TBFT**,SET,,,,COMP,PSHEAR).

Set the reference temperature at which your partial solution is performed (**TBFT**,SET,,,,TREF,TX). Only data at temperature TX is used to estimate shear coefficients.

Initialize the coefficients.

Solve (**TBFT**,SOLVE).

Repeat the above steps for all temperatures.

3. Solve the bulk coefficients (if any):

Set the partial solve option (**TBFT**,SET,,,,COMP,PBULK).

Set the reference temperature at which your partial solution is performed (**TBFT**,SET,,,,TREF,TX). Only data at temperature TX is used to estimate the shear coefficients.

Initialize the coefficients.

Solve (**TBFT**,SOLVE).

Repeat the above steps for all temperatures.

Alternatively, you can solve for both shear and bulk data at the same time:

1. Set the temperature dependency flag (**TBFT**,SET,,,,TDEP,1).
2. Solve for ALL coefficients.

Set the partial-solve option (**TBFT**,SET,,,,COMP,PVHE).

Set the reference temperature at which your partial solution is performed (**TBFT**,SET,,,,TREF,TX). Only data at temperature TX is used to estimate shear and bulk coefficients.

Initialize the coefficients.

Solve (**TBFT**,SOLVE).

Repeat the above steps for all of the desired temperatures.

6.2.6.4. Syntax and Arguments

The solution-option command is:

```
TBFT, SET, ID, CASE, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank -- not applicable)

Option4 = comp

Option5 = pshea (for Shear only) or pbulk (for bulk only)

The SOLVE option allows you to specify procedure types, tolerances, and number of iterations.

```
TBFT, SOLVE, ID, CASE, Option2, Option3, Option4, ..., Option7
```

where:

ID = Index corresponding to the material number

Option2 = visco function name (See [Table 6.4: Viscoelastic Data Types and Abbreviations](#) (p. 268).)

Option3 = (Blank--not applicable)

Option4 = Curve-fitting procedure: 0 = non-normalized least squares, 1 = normalized least squares (default)

Option5 = Maximum number of iterations

Option6 = Tolerance of residual changes

Option7 = Tolerance of coefficient changes

Other solving parameters are available (**TBFT**).

6.2.7. Step 6. Plot the Viscoelastic Experimental Data and Analyze

The best way to verify a good fit between your experimental data and the provided curves is to plot your curves and visually inspect them. The **Graph** button provides direct means to plot the data.

6.2.7.1. Analyze Your Curves for Proper Fit

All of your data is plotted as a function of column 1 for the X-axis. Columns 2 and above are each plotted in a separate graph, in a separate window, with the corresponding fitted data as a function of column 1. By default, all the shear data sets and/or the bulk data sets as well as the corresponding fitting results are plotted in two separated graphs. To view a specific data and its corresponding fitting result, click the right mouse button on the specific data set, and pick a desired option. You can also use the right mouse button to turn the legend and/or axis on and off. The scales can be also switched between log scale and regular scale. With the middle button, you can eliminate certain curves from each window's display in order to see the remaining data more clearly.

Right mouse button functions allow you to Zoom, Fit, Save Plot to File, View/Hide Legend, and View/Hide Grid. Two or more fitted functions can also be compared in the same plot. You can also see the number of points used to generate the plot, and also change the minimum X value and the maximum X value in a plot. Use the middle-mouse button (context sensitive) to hide a particular curve within a graph.

Reviewing your curve-fitting result graphically is the only way to ensure a good fit. After plotting the curve-fitting results, you can then review multiple plots and also verify the error norm/residual value that is printed in the curve-fitting window. This information helps you determine the quality of a curve fit and decide whether to accept the results. If not, you may want to go back to [step 3 \(p. 270\)](#), solve again by changing the order of the Prony series, redefining certain initial values of the coefficients, and possibly other control parameters. You can continue to use your original data, repeating [step 3 \(p. 270\)](#) through [step 6 \(p. 276\)](#) until you are satisfied with the solution.

6.2.8. Step 7. Write Viscoelastic Curve-Fitting Data to the Database

After a successful curve-fitting, the last step is to write the curve-fitting data as a Prony data table (**TB,PRONY**) to the database.

TBFT,FSET calculates coefficients that differ from those shown in the **TB,PRONY** table. The command uses the following equations to convert the calculated coefficients to the appropriate format:

Curve-Fitting Equation in Time Domain:

$$K = A_0^2 + \sum_{i=1-N} A_i^2 e^{-t/\tau_i}$$

Prony Equation:

$$K / K_{t=0} = \alpha_0 + \sum \alpha_i e^{-t/\tau_i}$$

α_i and τ_i ($i = 1$ to N) are the Prony coefficients (specified via **TB,PRONY**). A_0 is not entered in the Prony table.

Conversion Procedure:

$$K_{t=0} = \sum_{i=0-N} A_i^2$$

$$\alpha_i = \frac{A_i^2}{K_{t=0}}$$

Curve-Fitting Equation in Frequency Domain:

G' is the storage modulus, G'' is the loss modulus, Ω is the frequency in radians-per-unit-time, n is the order of the Prony model. A and τ are the fitted Prony parameters.

$$G' = A_0^2 + \sum_{i=1,n} \frac{A_i^2 (\tau_i \Omega)^2}{1 + (\tau_i \Omega)^2}$$

$$G'' = \sum_{i=1,n} \frac{A_i^2 \tau_i \Omega}{1 + (\tau_i \Omega)^2}$$

Conversion Procedure:

$$K_{t=0} = \sum_{i=0-N} A_i^2$$

$$\alpha_i = \frac{A_i^2}{K_{t=0}}$$

6.2.8.1. Syntax and Arguments

TBFT, FSET, ID, Option2, Option3

where:

Option2 = Case

Option3 = Case name

6.3. Chaboche Material Curve-Fitting

Chaboche material curve-fitting determines your material constants by relating your experimental data to the [Chaboche \(p. 51\)](#) nonlinear kinematic hardening model.

You can fit uniaxial plastic strain vs. stress data, along with discrete temperature dependencies for multiple data sets.

The following topics concerning Chaboche material curve-fitting are available:

- 6.3.1. Understanding the Chaboche Material Curve-Fitting Process
- 6.3.2. Step 1. Prepare Chaboche Experimental Data
- 6.3.3. Step 2. Input the Chaboche Experimental Data
- 6.3.4. Step 3. Select a Chaboche Material Model Option
- 6.3.5. Step 4. Initialize the Chaboche Coefficients
- 6.3.6. Step 5. Specify Chaboche Control Parameters and Solve
- 6.3.7. Step 6. Plot the Chaboche Experimental Data and Analyze
- 6.3.8. Step 7. Write Chaboche Curve-Fitting Data to the Database

6.3.1. Understanding the Chaboche Material Curve-Fitting Process

Chaboche material curve-fitting determines your material constants by relating your experimental data to the Chaboche nonlinear kinematic hardening model. Isotropic hardening can also be modeled by including a supported isotropic hardening model with the kinematic hardening model in the curve-fitting process.

Following is the general process for using curve-fitting to determine the coefficients for the Chaboche (p. 51) material model:

Step	Detailed Information Found Here	Comments
1	Step 1. Prepare Chaboche Experimental Data (p. 279)	The experimental data must be a plain text file with headers to describe the data types and attributes. The test data must be delimited by a space or a comma.
2	Step 2. Input the Chaboche Experimental Data (p. 280)	Read in the experimental data by specifying the file location on the command line.
3	Step 3. Select a Chaboche Material Model Option (p. 280)	The material options for the applicable curve-fitting regimen are defined (TBFT). This step includes selecting the kinematic hardening model.
4	Step 4. Initialize the Chaboche Coefficients (p. 281)	Chaboche curve-fitting is a nonlinear regression; the initial values of the coefficients to be determined is important for a successful solution.
5	Step 5. Specify Chaboche Control Parameters and Solve (p. 283)	Specify the error norm to be used, the solution control parameters, and perform the nonlinear regression.
6	Step 6. Plot the Chaboche Experimental Data and Analyze (p. 285)	Review and verify the results by comparing them with the experimental data and the regression errors. If any factor is unacceptable, repeat steps 3 through 5 to obtain a new curve-fitting solution.
7	Step 7. Write Chaboche Curve-Fitting Data to the Database (p. 286)	Write the curve-fitting results in the TB command format to the database.

6.3.2. Step 1. Prepare Chaboche Experimental Data

Curve fitting requires experimental test data. To use curve-fitting with plasticity, the only experimental data supported is uniaxial test data. Uniaxial test data has two columns, plastic strain and true stress. Experimental data for plasticity is path dependent.

Your uniaxial test data must be a plain text file with headers to define the test data. The data file should be in table format, delimited by spaces or commas. Headers can be used to describe the data types that characterize the test data columns or additional attributes of the data.

For Chaboche curve-fitting with multiple temperatures, you can evaluate coefficients at each discrete temperature point and write it as a temperature-dependent Chaboche data table. A separate data file is necessary for each discrete temperature.

Issue this command at the top of the experimental data file to specify the temperature for the experiment:

```
/temp,TempValue
```

where *TempValue* is your specified temperature.

Following is a typical data input file:

```
/temp,100 ! define temperature attribute
4.57E-06 2.43E+02
4.89E-04 2.67E+02
1.01E-03 2.83E+02
1.55E-03 2.94E+02
2.11E-03 3.02E+02
2.68E-03 3.09E+02
3.26E-03 3.14E+02
3.84E-03 3.18E+02
4.42E-03 3.22E+02
4.42E-03 7.78E+01
4.41E-03 -1.65E+02
3.54E-03 -2.31E+02
2.51E-03 -2.66E+02
1.40E-03 -2.84E+02
2.49E-04 -2.95E+02
-9.11E-04 -3.03E+02
-2.08E-03 -3.10E+02
-3.24E-03 -3.17E+02
-4.41E-03 -3.24E+02
-4.41E-03 -7.95E+01
-4.40E-03 1.63E+02
-3.53E-03 2.30E+02
-2.50E-03 2.65E+02
-1.39E-03 2.83E+02
-2.44E-04 2.94E+02
9.16E-04 3.02E+02
2.08E-03 3.09E+02
3.25E-03 3.16E+02
4.41E-03 3.23E+02
4.41E-03 7.87E+01
4.41E-03 -1.64E+02
3.53E-03 -2.31E+02
2.51E-03 -2.66E+02
1.40E-03 -2.84E+02
2.48E-04 -2.95E+02
```

Uniaxial test data can include loading, unloading, and cyclic loading.

For plasticity, experimental data is path-dependent and the stress-strain behavior depends on the history of the loading and/or unloading.

6.3.3. Step 2. Input the Chaboche Experimental Data

The experimental data must be read in from a plain text file (**TBFT**,EADD). Prepare this file as described in [Step 1. Prepare Experimental Data](#) (p. 279).

Each file is viewed as a data set, and can be a complete set of experimental test data or a part of a series of files of experimental test data. You can include several data sets, such as tests performed at different stress levels and/or temperatures, when you perform creep curve-fitting.

6.3.3.1. Syntax and Arguments

Issue this command to identify and specify the location of a data file:

```
TBFT, EADD, ID, Option1, Option2, Option3, Option4
```

where:

ID = Index corresponding to the material number

Option1 = Experimental data type UNIA (uniaxial test data)

Option2 = Experimental data file name

Option3 = File name extension

Option4 = File directory

6.3.4. Step 3. Select a Chaboche Material Model Option

TBFT provides the curve-fitting tool for Chaboche material modeling.

To define the material model, specify a case name, the order of Chaboche kinematic model, and the isotropic hardening option (if needed).

After creating a case, the number of Chaboche terms or the isotropic hardening options cannot be changed without deleting the case. You can create additional cases with different case names to model Chaboche kinematic hardening with different orders.

6.3.4.1. Syntax and Arguments

Define a Chaboche material model by defining a case (**TBFT**,FCASE), then specify the order of kinematic hardening model. The case is created only *after* **TBFT**,FCASE,ID,FINISH executes.

The following syntax example and argument descriptions illustrate a complete case definition:

```
TBFT, FCASE, ID, NEW, Option2, Option3 ! define case
```

where:

ID = Index corresponding to the material number

Option2 = CPLA

Option3 = Your specified case name

TBFT, FADD, ID, Category, Option2, Option3 ! specify kinematic hardening model

where:

ID = Index corresponding to the material number

Category = PLAS

Option2 = One of the following: CHABOCHE, BISO, MISO, VOCE, or POWER

Option3 = Dependent on *Option2*, as follows:

When *Option2* = CHABOCHE, *Option3* = 1 to N

When *Option2* = MISO, *Option3* = 1 to N_{iso}

When *Option2* = BISO, VOCE, or POWER, *Option3* is not used

TBFT, FCASE, ID, FINISH ! create case

6.3.5. Step 4. Initialize the Chaboche Coefficients

The initial values chosen for your [coefficients](#) (p. 28) (TBFT,SET) determines the success of the curve-fitting operation.

The number of parameter depends on the order of the Chaboche model. For a Chaboche model of order N, there are $2*N+1$ coefficients. Coefficients 1 through $2*N$ are Chaboche parameters, and coefficient $2*N+1$ is the yield stress. The odd coefficients in the Chaboche model refer to the slope of the curve and the even coefficients are decay function parameters. The number of terms depend on the complexity of the curve. Evenly distributed estimates of the slope over the range of the curve can be used as initial guesses for the curve-fitting process. Different terms can dominate at different parts of the curve. Initial guesses for the decay parameters can typically be chosen one or two orders less than odd coefficients (slope).

6.3.5.1. Including Isotropic Hardening Models with Chaboche Kinematic Hardening

When isotropic hardening models are included with Chaboche kinematic hardening, the number of coefficients is $2*N+N_{iso}$, where N_{iso} is:

- 2 for the BKIN option
- $2*N_i$ for the MISO option (where N_i is the number of MISO terms)
- 3 for the NLISO POWER option
- 4 for the NLISO VOCE option

The third coefficient for the power law is shear modulus, which must be set to the correct value and fixed before solving.

The index of the isotropic hardening coefficients start from $2*N+1$. Initial yield stress is generally estimated from separate experiments outside of curve-fitting and is generally fixed in the curve-fitting process.

For the BKIN, VOCE, and POWER options, the first isotropic hardening coefficient refers to yield stress.

6.3.5.2. General Process for Initializing MISO Option Coefficients

The MISO option used here is same as the **TB,PLAS,,,MISO** option. It has N_i coefficients, and the second isotropic hardening coefficient refers to the yield stress. Odd isotropic coefficients are plastic strain values and the even coefficients are stress values. The actual index of the coefficient is $2*N+N_{iso}$, where N is the order of the Chaboche model.

If the experiment has a maximum accumulated plastic strain of $\epsilon_{pl,max}$, set the odd coefficients to values equally distributed from 0 to $\epsilon_{pl,max}$. These values are then fixed (**TBFT,FIX**) before solving (**TBFT,SOLVE**). If you require greater accuracy at certain strain ranges, you can distribute the strain values unevenly as you wish.

To fix (hold constant) your coefficients (**TBFT,FIX**), specify a value for a coefficient and keep it unchanged, while allowing the other coefficients to be operated on. You can then release the fixed coefficient later if desired. By default, all of the coefficients are free to vary.

Estimate coefficients for temperature-dependent data by setting the temperature-dependency flag and setting a reference temperature before solving for the coefficients. You can set the reference temperature only to values specified via the **/temp, TempValue** header line (p. 279) in the experimental data.

You can also specify **TREF = ALL** to initiate multiple solves to evaluate coefficients at all available discrete temperature values. In this case, for data at three temperatures (t_1 , t_2 , and t_3), a single **TBFT,SOLVE** command initiates three separate solve operations at those three discrete temperature values, and generates data at three corresponding discrete temperatures.

With temperature dependency specified and the reference temperature set to a specific value, a **TBFT,SOLVE** command solves for coefficients only at that discrete temperature. To solve for coefficients at other temperatures, set the reference temperature to each of the desired discrete temperature values and solve again.

You can initialize the coefficients before or after activating temperature dependency. If the coefficients initialize before setting temperature dependency, the specified coefficients become the initial coefficients for all future solves for that particular model. These coefficients are, however, overridden when temperature dependency is active and another set of values is specified at a discrete temperature value. The curve-fitting tool looks for the initial coefficients at a particular temperature. If no coefficients are specified at discrete temperature values, the initial coefficients set before temperature dependency was activated are used.

6.3.5.2.1. Syntax and Arguments for Coefficient Initialization

```
TBFT, SET, ID, CASE, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = Index of coefficient

Option5 = Value of coefficient

Example 6.3: Initialize Coefficients

```
TBFT,SET,1,case,case1,,1,1.2      ! Initialize the first coefficient to 1.2
TBFT,SET,1,case,case1,,2,1.5      ! Initialize the second coefficient to 1.5
```

By default, [coefficients are not fixed \(p. 282\)](#). You can fix a coefficient to a value (set via **TBFT,SET**) or release a previously fixed coefficient (**TBFT,FIX**):

```
TBFT,FIX,ID,CASE,Option2,Option3,Option4,Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = Index of coefficient

Option5 = 1 to fix, 0 to vary (default)

Temperature dependency uses *Option4* and references your specified data files with the appropriate "temp" header:

```
TBFT,SET,ID,CASE,Option2,Option3,Option4,Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = TDEP or TREF

Option5 = If *Option4* = TDEP, then 1 activates temperature dependency 0 deactivates it.
If *Option4* = TREF, this value is either a specific temperature or ALL.

Example 6.4: Fixing Coefficient Values

```
TBFT,FIX,1,case,case1,,1,1      ! Fix the first coefficient to a value set via TBFT,SET
TBFT,FIX,1,case,case1,,2,1      ! Fix the second coefficient to a value set via TBFT,SET
```

6.3.6. Step 5. Specify Chaboche Control Parameters and Solve

Chaboche curve-fitting is a nonlinear regression process. Only the non-normalized error norm is available for the regression.

The solution control parameters for nonlinear regression include number of iterations, residual tolerance, and coefficient change tolerance. The solution stops when both residual tolerance of the error norm and coefficient change tolerance is met, or if the number of iterations criteria is met. The coefficients are updated when the solution is completed.

6.3.6.1. Chaboche Parameter Scaling

To facilitate convergence, you can activate parameter scaling (*Option9* = 1 on **TBFT,SOLVE**).

Chaboche parameters in general can be of different orders varying from around the initial stiffness in the stress- vs. plastic-strain curve to a very small value. Optimization algorithms used by the curve-fitting tool (**TBFT**) perform better when all parameters are scaled to a similar range.

Parameter scaling replaces C_i/Γ_i and Γ_i with $\text{EXP}(C_i')$ and $\text{EXP}(\Gamma_i')$, respectively, in the curve-fitting equations to improve the curve-fitting optimization process. The coefficients fit and estimated by the curve-fitting tool have different values during the curve-fitting process; however, the values are rescaled correctly when the parameter information is written in the appropriate **TB** table format to the database (**TBFT,FSET**).

6.3.6.2. Temperature-Dependent Solutions

Separate coefficients at each temperature must be calculated to account for temperature dependency. Perform the regression as follows:

1. Set the temperature-dependency flag (**TBFT,SET,,,,TDEP,1**).
2. Solve for all (TREF = ALL) coefficients.
 - a. Set the reference temperature at which your partial solution should be performed (**TBFT,SET,,,,TREF,TX**). Only data at temperature *TX* is used to estimate the combined plasticity coefficients.
 - b. Initialize the coefficients.
 - c. Solve (**TBFT,SOLVE**).

Repeat the regression process for all desired temperatures.

6.3.6.3. Syntax and Arguments

The following syntax examples and argument descriptions illustrate how to set control parameters and solve:

```
TBFT, SET, ID, CASE, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = TDEP

Option5 = 1 to activate temperature dependency, 0 to deactivate (default)

```
TBFT, SET, ID, CASE, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = TREF

Option5 = Valid temperature values found in the experimental data

The SOLVE option allows you to specify procedure types, tolerances, and the number of iterations:

```
TBFT, SOLVE, ID, CASE, Option2, Option3, Option4, ... , Option7
```

where:

ID = Index corresponding to the material number

Option2 = Case name

Option3 = (Blank--not applicable)

Option4 = Curve-fitting procedure: 0 = non-normalized least squares

Option5 = Maximum number of iterations

Option6 = Residual change tolerance

Option7 = Coefficient change tolerance

Other parameters for solving are available (**TBFT**).

6.3.7. Step 6. Plot the Chaboche Experimental Data and Analyze

The best method for ensuring a good fit between your experimental data and the provided curves is to plot your curves and visually inspect them. The **Graph** button provides a direct means for plotting the data.

6.3.7.1. Analyzing Your Curves for Proper Fit

All of your data is plotted as a function of column 1 (X axis). Column 2 (Y axis) and the corresponding fitted data are plotted as a function of column 1. Two or more fitted functions can be compared in the same plot.

Take advantage of the right-mouse-button functions to zoom, fit, save your plot to a file, view or hide objects, toggle between log scale and regular scale, and so on. With the middle mouse button, you can eliminate specific curves from each window's display in order to view the remaining data more clearly.

After plotting the curve-fitting results, you can then review multiple plots and also verify the error norm/residual value displayed in the curve-fitting window. This information helps you to determine the quality of a curve fit and decide whether or not to accept the results.

If the curve-fitting results are unsatisfactory, you may want to go back to [Step 3. Select a Material Model Option \(p. 280\)](#) and solve again by changing the order of the Chaboche model or other options, redefining certain initial values of the coefficients, and possibly redefining other control parameters. You can continue to use your original data, repeating [step 3 \(p. 280\)](#) through [step 6 \(p. 285\)](#) until you are satisfied with the solution.

6.3.8. Step 7. Write Chaboche Curve-Fitting Data to the Database

After a successful curve-fitting, the last step is to save the curve-fitting data as a Chaboche material data table (**TB,CHABOCHE**).

Issue **TBFT,FSET** to convert the coefficients in the appropriate **TB** table format and write them to the database.

6.4. Creep Material Curve-Fitting

Use curve-fitting to determine your creep material behavior. Thirteen creep models are available, along with the tools to generate and fit derived coefficients to your experimental data.

The following topics concerning creep material curve-fitting are available:

- 6.4.1. Understanding the Creep Material Curve-Fitting Process
- 6.4.2. Step 1. Prepare Creep Experimental Data
- 6.4.3. Step 2. Input the Creep Experimental Data
- 6.4.4. Step 3. Select a Creep Material Model Option
- 6.4.5. Step 4. Initialize the Creep Coefficients
- 6.4.6. Step 5. Specify Creep Control Parameters and Solve
- 6.4.7. Step 6. Plot the Creep Experimental Data and Analyze
- 6.4.8. Step 7. Write Creep Curve-Fitting Data to the Database
- 6.4.9. Hints for Curve-Fitting Creep Models

6.4.1. Understanding the Creep Material Curve-Fitting Process

The general process for creep curve-fitting is defined as follows:

Step	Detailed Information Found Here	Comments
1	Step 1. Prepare Creep Experimental Data (p. 287)	The experimental data must be a plain text file with headers to describe the data types and attributes. The test data must be delimited by a space or a comma.
2	Step 2. Input the Creep Experimental Data (p. 288)	Read the experimental data in by specifying the file location on the command line.
3	Step 3. Select a Creep Material Model Option (p. 289)	The material options for the applicable curve-fitting regimen are defined via TB . Support is offered for 13 implicit creep models. After you choose a model, you can still switch to another model if an ideal fit is not realized.
4	Step 4. Initialize the Creep Coefficients (p. 290)	Creep curve-fitting is a nonlinear regression; the initial values of the coefficients to be determined can be very important for a successful solution.
5	Step 5. Specify Creep Control Parameters and Solve (p. 291)	Choose the error norm to be used for an acceptable curve fit.

Step	Detailed Information Found Here	Comments
6	Step 6. Plot the Creep Experimental Data and Analyze (p. 293)	Review and verify the results by comparing them with the experimental data and the regression errors. If they are unacceptable, repeat steps 3 through 5 to obtain a new curve-fitting solution.
7	Step 7. Write Creep Curve-Fitting Data to the Database (p. 293)	Write the curve-fitting results in the TB command format to the database.

6.4.2. Step 1. Prepare Creep Experimental Data

You need to provide accurate experimental test data in order to achieve valid curve-fitting results. For creep analyses, you use either the creep strain value or creep strain rate, derived as a function of time, temperature, stress, and/or creep strain. The type of data required depends on the selected creep model. The experimental data is named "creep" to distinguish it from other types of data, such as uniaxial, tension, biaxial, etc. The creep data must be a plain text file that contains the headers and the test data in table form, delimited by a space or a comma.

The header is used to describe the data types that characterize the test data columns or attributes of the data. Five different creep data types are available:

Table 6.5: Creep Data Types and Abbreviations

Time	time
Equivalent Creep Strain	creq
Equivalent Creep Strain Rate	dcreq
Equivalent Stress	seqv
Temperature	temp

The header format to define each column's data type is `/n, abbr`, where *n* is the index of the data column in the file, and *abbr* is the abbreviation for the type of data in the column, as described in [Table 6.5: Creep Data Types and Abbreviations \(p. 287\)](#).

Following is a typical data input file:

```
/1,seqv ! indicates first column is stress
/2,creq ! indicates second column is creep strain
/3,temp ! indicates third column is temperature
/4,dcreq ! indicates fourth column is creep strain rate
4000 0.00215869 100 0.000203055
4000 0.00406109 100 0.000181314
4000 0.00664691 100 0.000165303
4000 0.0102068 100 0.000152217
4000 0.0151416 100 0.000140946
```

When a particular column is unchanged over the loading history, you can define it as an attribute. For instance, in the above example, the stress and temperature are constant throughout the range. You define this data as an attribute.

The header format to define a data attribute is `/attr, value`, where *attr* is the data-type abbreviation, and *value* is the value of the attribute. The constant stress and temperature values above can be written into the file header, as follows:

```

/seqv,4000 ! indicate this creep has a constant stress of 4000
/temp,100 ! indicate this creep data is at a constant temperature of 100
/1,creq ! indicate first column is creep strain
/2,dcreq ! indicate second column is creep strain rate
0.00215869 0.000203055
0.00406109 0.000181314
0.00664691 0.000165303
0.0102068 0.000152217
0.0151416 0.000140946
0.0220102 0.000130945

```

Thirteen model types are available for creep curve-fitting. The model you choose determines the experimental data required for the curve-fitting process. The following table describes the creep data required to perform curve-fitting for each model type. For strain hardening and modified strain hardening, you must input both creep strain and creep strain rate in the experimental data.

Table 6.6: Creep Model and Data/Type Attribute

Creep Model	creq	dcreq	time	seqv	temp
Strain Hardening	x	x		x	x
Time Hardening		x	x	x	x
Generalized Exponential		x	x	x	x
Generalized Graham		x	x	x	x
Generalized Blackburn		x	x	x	
Modified Time Hardening	x		x	x	x
Modified Strain Hardening	x	x		x	x
Generalized Garofalo		x		x	x
Exponential Form		x		x	x
Norton		x		x	x
Combined Time Hardening	x		x	x	x
Prim+Sec Rational Polynomial		x	x	x	
Generalized Time Hardening	x		x	x	x

6.4.3. Step 2. Input the Creep Experimental Data

The experimental data must be read in from a plain text file. Prepare this file according to the previous section, including both header information and formatted test data. The header portion is required for creep analyses.

Each file is viewed as a data set, and can be a complete set of experimental test data or a part of a series of files of experimental test data. You can include several data sets, such as tests performed at different loading conditions, strain ranges, and temperatures in separate files. Each file can have only one header.

6.4.3.1. Syntax and Arguments

TBFT,EADD identifies and specifies the location of your data files:

```
TBFT,EADD,ID,Option1,Option2,Option3,Option4
```

where:

ID = Index corresponding to the material number

Option1 = Creep

Option2 = Name of file containing experimental data

Option3 = File name extension

Option4 = File directory

6.4.4. Step 3. Select a Creep Material Model Option

The models available for creep curve-fitting are defined in [Table 6.7: Creep Models and Abbreviations \(p. 289\)](#). Select the one that best satisfies your requirements, and use the creep model abbreviation from the table in subsequent fitting operations. It is helpful to view the formula before solving, as you can better determine the initial coefficients to use and the format of your experimental data.

See [Table 6.7: Creep Models and Abbreviations \(p. 289\)](#) to determine a starting point for the initial creep model coefficients.

6.4.4.1. Syntax and Arguments

```
TBFT, FADD, ID, CREEP, Option2, Option3
```

where:

ID = Index corresponding to the material number

Option2 = creep model abbreviation. See [Table 6.7: Creep Models and Abbreviations \(p. 289\)](#).

Option3 = not used for creep curve-fitting.

The following table describes the creep models available and their abbreviated names for *Option2* (above).

Table 6.7: Creep Models and Abbreviations

Model Number	Name	Fitting Name/Option2
1	Strain Hardening	shar
2	Time Hardening	thar
3	Generalized Exponential	gexp
4	Generalized Graham	ggra
5	Generalized Blackburn	gbla
6	Modified Time Hardening	mtha
7	Modified Strain Hardening	msha
8	Generalized Garofalo	ggar
9	Exponential Form	expo
10	Norton	nort
11	Prim+Sec Time Hardening	psth
12	Prim+Sec Rational Polynomial	psrp
13	Generalized Time Hardening	gtha

The experimental data must be consistent with the creep model you choose. See [Table 6.6: Creep Model and Data/Type Attribute \(p. 288\)](#) for the data types required for each creep model.

6.4.5. Step 4. Initialize the Creep Coefficients

Creep curve-fitting is a nonlinear regression process. A successful curve fit depends greatly on the initial coefficient values; certain variances prevent your curve fit from converging. When a solution doesn't converge, you can adjust the initial value of specific coefficients and rerun the problem (you do not need to input the data again). In general, the more parameters a model has, the more difficult it is to get the solution to converge.

Models with many parameters sometimes converge more easily if you fix (hold constant) your coefficients. You specify a value for a coefficient and keep it unchanged, while allowing the other coefficients to be operated on. You can then release the fixed coefficients to obtain a solution. By default, all of the coefficients are free to vary.

Although some creep models implicitly address temperature dependency, other models lack this capability. For both types of models, you can account for temperature dependency by generating coefficients at discrete temperature values.

Solve for separate coefficients at each of your desired discrete temperatures. If the equation implicitly supports temperature dependency already, that portion is eliminated by appropriately fixing the coefficient to a specific value. For example, if you have $e(-C*T)$, then C is set to 0 to eliminate this term. To do this, you activate temperature dependency by setting $TDEP = 1$.

For temperature-dependent experimental data, you enable temperature-dependency and specify a reference temperature before solving for the coefficients. You can set the reference temperature only to values specified using the `/temp,value` header line in the experimental data.

You can also specify $TREF = ALL$ and initiate multiple solves to evaluate coefficients at all available discrete temperature values. In this case, for data at three temperatures (t_1 , t_2 , and t_3), a single **TBFT,SOLVE** entry initiates three different solve operations at those three discrete temperatures.

With temperature dependency on and the reference temperature set to a particular value, a **TBFT,SOLVE** command solves for coefficients only at that temperature. To solve for coefficients at other temperatures, you set the reference temperature to each of the desired discrete temperature values and solve again.

You can initialize the coefficients before or after turning temperature dependency on. If the coefficients are initialized before turning temperature dependency on, the specified coefficients become the initial coefficients for all future solves for that particular model. These coefficients are, however, overridden when temperature dependency is turned on and another set of initial values is specified at discrete temperatures. The curve-fitting tool looks for the initial coefficients at a particular temperature. If no coefficients are specified at discrete temperatures, the initial coefficients set before temperature dependency was turned on are used.

6.4.5.1. Syntax and Arguments

TBFT,SET defines coefficient values and temperature dependency:

```
TBFT, SET, ID, CREEP, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = creep model name

Option3 = order, if applicable

Option4 = index of coefficient

Option5 = value of coefficient

TBFT, FIX modifies the coefficients:

```
TBFT, FIX, ID, CREEP, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = creep model name

Option3 = order, if applicable

Option4 = index of coefficient

Option5 = 0 - variable, 1 - fixed

```
TBFT, SET, ID, CREEP, Option2, Option3, Option4, Option5
```

where:

ID = Index corresponding to the material number

Option2 = creep model name

Option3 = order, if applicable

Option4 = TDEP or TREF

Option5 = For *Option4* = TDEP, 1 to activate, 0 to deactivate. For *Option4* = TREF, a specific temp value, or ALL.

6.4.6. Step 5. Specify Creep Control Parameters and Solve

Your error norm, maximum number iterations allowed, and error tolerance affect the accuracy of your results. Two error norms are available for the regression. The default error norm calculation option is normalized curve-fitting. It usually yields better results than non-normalized curve-fitting, as normalized fitting gives equal weight to all data points when minimizing the error norms.

Other available solve criteria are number of iterations, residual tolerance, and coefficient change tolerance. The solution stops when both residual tolerance and coefficient change tolerance is met or if the number of iterations criteria is met. The coefficients are updated after every iteration during the solve process.

The following topics for specifying control parameters and solving are available:

[6.4.6.1. Syntax and Arguments](#)

[6.4.6.2. Creep Parameter Scaling](#)

6.4.6.1. Syntax and Arguments

```
TBFT, SOLVE, ID, CREEP, Option2, Option3, Option4, ..., Option7
```

where:

ID = Index corresponding to the material number

Option2 = creep function name (See [Table 6.7: Creep Models and Abbreviations \(p. 289\)](#))

Option3 = ((Blank--not applicable for creep models)

Option4 = Error norm: 0 = non-normalized, 1 = normalized (default)

Option5 = Maximum number of iterations

Option6 = Tolerance of residual changes

Option7 = Tolerance of coefficient changes

Other solution parameters are available (**TBFT**).

6.4.6.2. Creep Parameter Scaling

To facilitate convergence, you can activate parameter scaling (*Option9* = 1 on **TBFT,SOLVE**).

Parameters for creep curve-fitting can have a wide range of values. For example, C1 in the exponential form creep option is of the same order of strain, whereas the C2 parameters can be a number between 0 and 5. Optimization algorithms used by the curve-fitting tool perform better when all parameters are scaled to a similar range.

The coefficients fit and estimated by the curve-fitting tool have different values during the curve-fitting process; however, the values are rescaled correctly when the parameter information is written in the appropriate **TB** table format to the database (**TBFT,FSET**).

Creep coefficients are initialized automatically (unless you have issued **TBFT,SET**) with a default set of initial coefficients based on the experimental data. You can determine whether the coefficients produce a good fit (**TBFT,SOLVE**), then experiment with the initialization or different models if the fits are unsatisfactory. Parameter scaling with automatic initialization improves the curve-fitting process.

The following table shows how parameter scaling modifies the parameter values to provide a more evenly scaled parameter set. For example, C1 is replaced by EXP(C1) in the fitting equations, so that C1 is no longer a tiny number when compared to the other parameters C2, C3, and C4.

Creep Option	Creep Model Name	Parameters									
1	Strain Hardening	EXP(C1)	C2	C3	C4						
2	Time Hardening	EXP(C1)	C2	C3	C4						
3	Generalized Exponential	EXP(C1)	C2	C3	C4	EXP(C5)					
4	Generalized Graham	EXP(C1)	C2	C3	C4	C5	C6	C7	C8		
5	Generalized Blackburn	EXP(C1)	C2	C3	C4	C5	EXP(C6)	C7			
6	Modified Time Hardening	EXP(C1)	C2	C3	C4						
7	Modified Strain Hardening	EXP(C1)	C2	C3	C4						

Creep Option	Creep Model Name	Parameters									
8	Generalized Garofalo	EXP(C1)	C2	C3	C4						
9	Exponential form	EXP(C1)	C2	C3							
10	Norton	EXP(C1)	C2	C3							
11	Combined Time Hardening	EXP(C1)	C2	C3	C4	EXP(C5)	C6	C7			
12	Rational polynomial	EXP(C1)	C2*C2	C3	C4	C7	C8	C9	C10	C11	C12
13	Generalized Time Hardening	EXP(C1)	EXP(C2)	EXP(C3)	C4	C5	C6				

6.4.7. Step 6. Plot the Creep Experimental Data and Analyze

At solution, the coefficient tables contain the fitted coefficients and also the residual errors. You can then plot your data and visually interpret the results. Column one is your X- axis, with each additional column plotted separately as a function of column one.

Reserve column one for the variable that you would like to see vary in the plot. For example, if your data contains time, temperature, stress and creep strain, you may wish to see the creep strain vary as a function of time, at different temperatures and stresses in the plot. Add your experimental data (via multiple **TBFT**, **EADD** commands). Split the file into multiple experimental fields as prescribed earlier, one for each combination of temperature and stress.

Right mouse button (RMB) functions allow you to Zoom, Fit, Save Plot to File, View/Hide Legend and View/Hide Grid. Two or more fitted functions can also be compared in the same plot. For example, you can view Mooney2 Uniaxial and Mooney9 Uniaxial plots right on top of each other, provided both of these function are already solved for. RMB also allows you to see the number of points used to generate the plot, and also change the Minimum X Value and the Maximum X Value in a plot. You can use the middle-mouse button (context sensitive) to hide a particular curve within a graph.

6.4.7.1. Analyze Your Curves for Proper Fit

Graphically review the curve-fitting result and ensure a good fit throughout the range of data. Use the plotted curve-fitting results both to determine the degree of fit at various locations and to verify the error norm/residual value. You can then determine the quality of a curve-fitting and decide whether to accept the results. If the results are unacceptable, you may want to go back to [Step 3 \(p. 289\)](#), then solve again with a new model, redefining certain initial values of the coefficients, and also modifying some of the other control parameters. You can continue to use your experimental data, repeating [Step 3 \(p. 289\)](#) through [Step 6 \(p. 293\)](#) until you obtain a satisfactory solution.

6.4.8. Step 7. Write Creep Curve-Fitting Data to the Database

When you are satisfied with the curve-fitting results, write the curve-fitting data to the database (**TBFT**, **FSET**). The program converts the coefficients to the appropriate **TB** data table format before writing to the database, storing the data as part of the material property set for use in subsequent analyses.

6.4.8.1. Syntax and Arguments

```
TBFT, FSET, ID, CREEP, Option2, Option3
```

where:

ID = Index corresponding to the material number

Option2 = Creep model abbreviation

Option3 = (Blank--not applicable)

6.4.9. Hints for Curve-Fitting Creep Models

The following tips can help to ensure successful curve-fitting. They are only suggestions, and do not represent a singular method or strategy. Also, following them does not guarantee a solution. Refer to [Creep \(p. 81\)](#) for additional details about each implicit creep model.

Strain Hardening

Strain hardening has four coefficients, with C4 dedicated to temperature dependency. If you do not have temperature-dependent data, set C4 to zero. If you have difficulty solving temperature-dependent data, use experimental data for only one temperature and fix C4 to zero. Solve, and then add data for your other temperatures. Then release C4 and solve for all coefficients. You can also solve for just C4 by fixing the C1, C2 and C3 values.

Time Hardening

Time hardening has four coefficients, with C4 dedicated to temperature dependency. If you do not have temperature-dependent data, set C4 to zero. If you have difficulty solving temperature-dependent data, use experimental data for only one temperature and fix C4 to zero. Solve, and then add data for your other temperatures. Then release C4 and solve for all coefficients. You can also solve for just C4 by fixing the C1, C2 and C3 values.

Generalized Exponential

Generalized exponential has five coefficients, with C4 dedicated to temperature dependency. If you have no temperature-dependent data, set C4 to zero. Use a low value for C5 (such as 1e-3) to avoid floating-point overflows in the exponential term. If you have difficulty solving for temperature-dependent data, use experimental data for only one temperature, fix C4 to zero, then solve. Then add data for your other temperatures, release C4 and solve for all coefficients. You can also solve for just C4 by fixing C1, C2 and C3.

Generalized Graham

Generalized Graham has eight coefficients. You use C8 for temperature dependency. If you do not have temperature-dependent data, set C8 to zero. If you have difficulty solving temperature-dependent data, use experimental data for only one temperature, fix C8 to zero, and solve. Then add data for other temperatures, release C8 and solve for the remaining coefficients individually.

Generalized Blackburn

Generalized Blackburn has seven coefficients. Examine the exponential terms and try to prevent floating-point overflows. To keep $e^{C_2\sigma}$ within floating-point range, ensure that the initial value of C2 is such that $C_2\sigma$ is close to 1. Similarly, try to keep σ/C_4 and $C_7\sigma$ close to 1.

Modified Time Hardening	Modified time hardening has four coefficients. C4 is for temperature dependency. If you do not have temperature-dependent data, fix C4 to zero.
Modified Strain Hardening	Modified Strain Hardening has three coefficients. This model can be complex for curve-fitting. To keep the $C_1\sigma^{C_2}[(C_3+1)\varepsilon]^{C_3}$ term from going negative, C1 is replaced with C_1^2 but converted to the proper form before being written to the database.
Generalized Garofalo	Generalized Garofalo has four coefficients, with C4 dedicated to temperature dependency. If you do not have temperature-dependent data, set C4 to zero. To keep the Sinh term within floating-point range, keep $c_2\sigma$ close to one when you initialize the coefficients.
Exponential Form	Exponential form has three coefficients, with C3 dedicated to temperature dependency. If you do not have temperature-dependent data, set C3 to zero. To keep e^{σ/C_2} within floating-point range, keep σ/C_2 close to one.
Norton	Norton model has three coefficients, with C3 dedicated to temperature dependency. If you do not have temperature-dependent data, set C3 to zero.
Prim+Sec Time Hardening	Time hardening has seven coefficients. This is a complex model. Here it is advisable to solve for temperature independent data first and then introduce temperature related data. To keep the coefficients positive and to improve convergence, the curve-fitting process replaces coefficients C1 and C5 with $\exp(C_1)$ and $\exp(c_5)$, respectively, and converts them appropriately before generating the material data tables.
Prim+Sec Rational Polynomial	Rational polynomial is a very complex model for curve-fitting, with 10 coefficients. If you find it difficult to fit this data, try splitting the experimental data into primary and secondary creep data. Primary creep data is the initial part of the curve that covers the nonlinearity in the strain rate. Fit only the secondary data by fixing C1 to 1 and then set all other coefficients except C2, C3 and C4 to zero. Use a low value of C3 to keep $10^{C_3\sigma}$ within floating-point range. Coefficients C5 to C10 in curve-fitting refers to coefficients C7 to C12 in the implicit creep equation (rational polynomial) (p. 82). Then add the primary creep data, release all coefficients, and solve.
Generalized Time Hardening	Generalized time hardening has six coefficients. Set C6 to zero if you have temperature independent data. When initializing coefficients set $C_5\sigma$ close to 1 to avoid floating-point overflows.

Chapter 7: Material Model Combinations

You can combine some material models to simulate various material behaviors. The following table presents the model options you can combine along with the associated **TB** command labels, and links to sample input listings located under [Material Model Combinations](#) in the *Structural Analysis Guide*. If a material model does not appear in the table, it cannot be combined with another material model unless specifically noted elsewhere.

Table 7.1: Material Model Combination Possibilities

Model	With ...	Combination Type	Command, Label	Link to Example
Plasticity	Combined Hardening	Bilinear	TB ,BISO + TB ,CHAB	BISO and CHAB Example
Plasticity	Combined Hardening	Multilinear	TB ,PLAS,,,,MISO + TB ,CHAB	PLAS (MISO) and CHAB Example
Plasticity	Combined Hardening	Nonlinear	TB ,NLISO + TB ,CHAB	NLISO and CHAB Example
Plasticity	Combined Hardening	Multilinear	TB ,PLAS,,,,MISO + TB ,EDP	PLAS (MISO) and EDP Example
Viscoplasticity	Isotropic Hardening	Bilinear	TB ,BISO + TB ,RATE	BISO and RATE Example
Viscoplasticity	Isotropic Hardening	Multilinear	TB ,PLAS,,,,MISO + TB ,RATE	PLAS (MISO) and RATE Example
Viscoplasticity	Isotropic Hardening	Nonlinear	TB ,NLISO + TB ,RATE	NLISO and RATE Example
Viscoplasticity	Combined Hardening	Nonlinear	TB ,CHAB + TB ,RATE + TB ,BISO	CHAB and RATE and BISO Example
Viscoplasticity	Combined Hardening	Nonlinear	TB ,CHAB + TB ,RATE + TB ,PLASTIC	CHAB and RATE and PLASTIC Example
Viscoplasticity	Combined Hardening	Nonlinear	TB ,CHAB + TB ,RATE + TB ,PLAS + TB ,PLAS,,,,KSR2	CHAB and RATE and PLASTIC and KSR2 Example
Viscoplasticity	Combined Hardening	Nonlinear	TB ,CHAB + TB ,CREEP + TB ,RATE + TB ,PLAS	CHAB and CREEP and RATE and PLASTIC Example
Viscoplasticity	Combined Hardening	Nonlinear	TB ,CHAB + TB ,CREEP + TB ,RATE + TB ,PLAS + TB ,PLAS,,,,KSR2	CHAB and CREEP and RATE and PLASTIC and KSR2 Example
Viscoplasticity	Combined Hardening	Nonlinear	TB ,CHAB+ TB ,RATE+ TB ,NLISO	CHAB and RATE and NLISO Example

Model	With ...	Combination Type	Command, Label	Link to Example
Gurson Plasticity	Isotropic Hardening	Bilinear	TB,GURS + TB,BISO	GURSON and BISO Example
Gurson Plasticity	Isotropic Hardening	Multilinear	TB,GURS + TB,PLAS,,,MISO	GURSON and PLAS (MISO) Example
Gurson Plasticity	Isotropic Hardening	Nonlinear	TB,GURS + TB,NLISO	GURSON and NLISO Example
Gurson Plasticity	Isotropic Hardening	Nonlinear	TB,GURS + TB,CHAB	GURSON and CHAB Example
Gurson Plasticity	Isotropic Hardening	Nonlinear	TB,GURS + TB,CHAB + TB,BISO or NLISO , or TB,PLAS,,,MISO	GURSON and CHAB and BISO Example
Plasticity and Creep (Implicit)	Isotropic Hardening	Bilinear	TB,BISO + TB,CREEP	BISO and CREEP Example
Plasticity and Creep (Implicit)	Isotropic Hardening	Multilinear	TB,PLAS,,,MISO + TB,CREEP	PLAS (MISO) and CREEP Example
Plasticity and Creep (Implicit)	Isotropic Hardening	Nonlinear	TB,NLISO + TB,CREEP	NLISO and CREEP Example
Plasticity and Creep (Implicit)	Kinematic Hardening	Bilinear	TB,BKIN + TB,CREEP	BKIN and CREEP Example
Plasticity and Creep (Implicit)	Kinematic Hardening	Nonlinear	TB,CHAB + TB,CREEP	CHAB and CREEP Example
Plasticity and Creep (Implicit)	Kinematic Hardening	Nonlinear	TB,CHAB + TB,PLAS,,,KSR2 + TB,CREEP	CHAB and PLAS and CREEP Example
Anisotropic Plasticity	Isotropic Hardening	Bilinear	TB,HILL + TB,BISO	HILL and BISO Example
Anisotropic Plasticity	Isotropic Hardening	Multilinear	TB,HILL + TB,PLAS,,,MISO	HILL and PLAS (MISO) Example
Anisotropic Plasticity	Isotropic Hardening	Nonlinear	TB,HILL + TB,NLISO	HILL and NLISO Example
Anisotropic Plasticity	Kinematic Hardening	Bilinear	TB,HILL + TB,BKIN	HILL and BKIN Example
Anisotropic Plasticity	Kinematic Hardening	Multilinear	TB,HILL + TB,PLAS,,,KINH	HILL and PLAS (Kinematic Hardening) Example
Anisotropic Plasticity	Kinematic Hardening	Chaboche	TB,HILL + TB,CHAB	HILL and CHAB Example
Anisotropic Plasticity	Combined Hardening	Bilinear Isotropic and Chaboche	TB,HILL + TB,BISO + TB,CHAB	HILL and BISO and CHAB Example

Model	With ...	Combination Type	Command, Label	Link to Example
Anisotropic Plasticity	Combined Hardening	Multilinear Isotropic and Chaboche	TB ,HILL + TB ,PLAS,,,,MISO + TB ,CHAB	HILL and PLAS (MISO) and CHAB Example
Anisotropic Plasticity	Combined Hardening	Nonlinear Isotropic and Chaboche	TB ,HILL + TB ,NLISO + TB ,CHAB	HILL and NLISO and CHAB Example
Anisotropic Viscoplasticity	Isotropic Hardening	Bilinear	TB ,HILL + TB ,RATE + TB ,BISO	HILL and RATE and BISO Example
Anisotropic Viscoplasticity	Isotropic Hardening	Nonlinear	TB ,HILL + TB ,RATE + TB ,NLISO	HILL and RATE and NLISO Example
Anisotropic Creep (Implicit)	---		TB ,HILL + TB ,CREEP	HILL and CREEP Example
Anisotropic Creep and Plasticity (Implicit)	Isotropic Hardening	Bilinear	TB ,HILL + TB ,CREEP + TB ,BISO	HILL and CREEP and BISO Example
Anisotropic Creep and Plasticity (Implicit)	Isotropic Hardening	Nonlinear	TB ,HILL + TB ,CREEP + TB ,NLISO	HILL and CREEP and NLISO Example
Anisotropic Creep and Plasticity (Implicit)	Kinematic Hardening	Bilinear	TB ,HILL + TB ,CREEP + TB ,BKIN	HILL and CREEP and BKIN Example
Anisotropic Creep and Plasticity (Implicit)	Kinematic Hardening	Chaboche	TB ,HILL + TB ,CHAB + TB ,CREEP	HILL and CHAB and CREEP Example
Visco-Anisotropic Hyperelasticity		Nonlinear Anisotropic Elasticity	TB ,AHYPER + TB ,PRONY	AHYPER and PRONY Example
Visco-Hyperelasticity		Nonlinear Isotropic Elasticity	TB ,HYPER + TB ,PRONY	HYPER and PRONY Example
Visco-Hyperelasticity	Mullins Effect	Nonlinear Isotropic Elasticity	TB ,HYPER + TB ,PRONY + TB ,CDM	HYPER and PRONY and CDM Example
Hyperelasticity	Mullins Effect	Nonlinear Isotropic Elasticity	TB ,HYPER + TB ,CDM	HYPER and CDM Example
Cap and Creep (Implicit)	Isotropic Hardening	Bilinear or Nonlinear	TB ,EDP + TB ,CREEP + TB ,BISO or TB ,NLISO	CAP and CREEP and PLAS (MISO) Example

Model	With ...	Combination Type	Command, Label	Link to Example
Cast Iron	Nonlinear Kinematic Hardening	Nonlinear	TB ,CAST + TB ,CHAB	CAST and CHAB Example

Following are cross-reference links to other sections in the documentation that provide descriptions of the *individual* material model options represented in the table above.

- [Bilinear Isotropic Hardening \(p. 40\)](#) (**TB**,BISO)
- [Bilinear Kinematic Hardening \(p. 47\)](#) (**TB**,BKIN)
- [Chaboche Nonlinear Kinematic Hardening \(p. 51\)](#) (**TB**,CHAB)
- [Creep \(Implicit\) \(p. 81\)](#) (**TB**,CREEP) -- Also see [Creep](#).
- [Hill Anisotropy \(p. 38\)](#) (**TB**,HILL]
- [Nonlinear Isotropic Hardening \(p. 43\)](#) (**TB**,NLISO).
- [Rate-Dependent Plasticity \(p. 78\)](#) (**TB**,RATE)
- [Hyperelasticity \(p. 83\)](#) (**TB**,HYPER)
- [Anisotropic Hyperelasticity \(p. 93\)](#) (**TB**, AHYPER) -- Also see [Anisotropic Hyperelasticity](#).
- [Viscoelasticity](#)
- [Cast Iron \(p. 69\)](#) (**TB**,CAST)

Chapter 8: Understanding Field Variables

When you define field-dependent material properties (**TBFIELD** and related **TB** commands), the program uses linear interpolation between the data points you provide to determine specific material property values. To do so, the program:

1. Creates a grid using your defined field data values.
2. Assumes that you have defined curve-based data and automatically provides the missing grid data points.
3. Performs linear interpolation on this populated grid to find material property values.
4. Enables editing of the initialized field variables during solution, if necessary.

The following field-variable topics are available:

- 8.1. [Predefined Field Variables](#)
- 8.2. [User-Defined Field Variables](#)
- 8.3. [Data Processing](#)
- 8.4. [Logarithmic Interpolation and Scaling](#)
- 8.5. [Interpolation Algorithms](#)

8.1. Predefined Field Variables

The following predefined field variables (defined via **TBFIELD**) are supported by the material models indicated:

Table 8.1: Predefined Field Variables and Supported Material Data Tables

Field Variable	Description	Material Model (TB , <i>Lab</i>) Support
TEMP	Temperature	All material models via TBTEMP . If using the material models listed in this table, you can also define temperature via TBFIELD ,TEMP.
FREQ	Frequency	• Elasticity (TB,ELASTIC) • Structural damping (TB,SDAMP) • Structural damping (Rayleigh) (p. 227) (TB,SDAMP, where <i>TBOPT</i> = ALPD / BETD) • Thermal expansion (p. 21) (TB,CTE) • Perforated material (TB,PERF) • Acoustic material (TB,AFDM)

Field Variable	Description	Material Model (TB,Lab) Support
CYCLE	Healing cycle	Cohesive Zone Material (p. 209) (TB,CZM)
NPRES	Normal pressure	Coefficient of friction (p. 204) (TB,FRIC)
SLDI	Sliding distance	
SLRV	Sliding velocity	
TIME	Solution time	- Creep (p. 81) (TB,CREEP) - Elasticity (TB,ELASTIC) - Extended Drucker-Prager (p. 55) (TB,EDP) - Hyperelasticity (p. 83) (TB,HYPER) - Mass density (TB,DENS) - Plasticity (TB,PLASTIC with isotropic option [TB OPT = MISO]) - Chaboche kinematic hardening (TB,CHABOCHE) - Rate-dependent plasticity (p. 78) (TB,RATE) - Gurson plasticity (p. 64) (TB,GURSON) - Thermal expansion (p. 21) (TB,CTE) - Thermal properties (p. 257) (TB,THERM) [2] - Structural damping (Rayleigh) (p. 227) (TB,SDAMP, where TB OPT = ALPD / BETD) - Coefficient of friction (p. 204) (TB,FRIC) - Contact surface wear (TB,WEAR)
UX, UY, UZ [1]	Displacements in the global/local X, Y, or Z coordinate system, respectively	- Creep (p. 81) (TB,CREEP) - Elasticity (TB,ELASTIC) - Extended Drucker-Prager (p. 55) (TB,EDP) - Hyperelasticity (p. 83) (TB,HYPER) - Plasticity (TB,PLASTIC with isotropic option [TB OPT = MISO]) - Rate-dependent plasticity (p. 78) (TB,RATE) - Thermal expansion (p. 21) (TB,CTE)

Field Variable	Description	Material Model (TB,Lab) Support
XCOR, YCOR, ZCOR	Global or local coordinate system locations	• Creep (p. 81) (TB,CREEP) • Elasticity (TB,ELASTIC) • Extended Drucker-Prager (p. 55) (TB,EDP) • Hyperelasticity (p. 83) (TB,HYPER) • Mass density (TB,DENS) • Plasticity (TB,PLASTIC with isotropic option [TBOPT = MISO]) • Chaboche kinematic hardening (TB,CHABOCHE) • Rate-dependent plasticity (p. 78) (TB,RATE) • Gurson plasticity (p. 64) (TB,GURSON) • Thermal expansion (p. 21) (TB,CTE) • Thermal properties (p. 257) (TB,THERM) [2]
UF01 - UF09	User-defined (p. 304)	
PPRE	Pore-pressure degree of freedom	• Elasticity (TB,ELASTIC) • Extended Drucker-Prager (p. 55) (TB,EDP) • Hyperelasticity (p. 83) (TB,HYPER) • Mass density (TB,DENS) • Plasticity (TB,PLASTIC with isotropic option [TBOPT = MISO]) • Chaboche kinematic hardening (TB,CHABOCHE) • Thermal expansion (p. 21) (TB,CTE) • Thermal properties (p. 257) (TB,THERM) [2]

1. For elements with displacement degrees of freedom only.
2. Coupled-pore-pressure-thermal elements (CPTnnn) not supported.

The reference coordinate systems for location (XCOR, YCOR, ZCOR) and displacement (UX,UY,UZ) can be modified from global (default) to any user-defined Cartesian coordinate system specified via **TBEO**. Coordinate transformation includes both translation and rotation; if you require only rotation for displacement, define a coordinate system that only has rotation.

In a linear analysis, when a field variable changes during solution, the full Newton-Raphson option (**NROPT,FULL**) is required to obtain the correct behavior.

8.1.1. Defining Friction

The TEMP value corresponds to the average temperature on the contact surface for contact elements [CONTA172](#), [CONTA174](#), [CONTA175](#), and [CONTA177](#). For contact element [CONTA178](#), the TEMP value corresponds to the average temperature of the nodes.

The TIME value corresponds to the analysis time specified via [TIME](#).

The algebraic sliding distance SLDA is the total sliding distance (the algebraic sum) as reported in the element output definitions table for the contact elements (for example, TASS and TASR output items for [CONTA174](#)).

The absolute sliding distance SLDI is the total accumulated sliding distance (the absolute sum) as reported in the element output definitions table for the contact elements (for example, AASS and AASR output items for [CONTA174](#)).

When used with [TB,FRIC](#), field variables defined via [TBFIELD](#) are available only for isotropic friction ($TBOPT = ISO$) and orthotropic friction ($TBOPT = ORTHO$ or $EORTHO$); they are not available for user-defined friction ($TBOPT = USER$).

For more information about using [TBFIELD](#) with [TB,FRIC](#), see [Contact Friction in the Material Reference](#) (p. 204).

8.1.2. Defining Young's Modulus as a Function of Global X,Y

```
! Young's Modulus as a Function of Global X,Y
TB,ELASTIC,1

TBFIELD,XCOR,0.0
TBFIELD,YCOR,1.0
TBDATA,1,1e6,0.3
TBFIELD,YCOR,5.0
TBDATA,1,1e7,0.3

TBFIELD,XCOR,1.0
TBFIELD,YCOR,1.0
TBDATA,1,1.1e6,0.3
TBFIELD,YCOR,5.0
TBDATA,1,1.1e7,0.3
```

8.2. User-Defined Field Variables

User-defined field variables ([TBFIELD](#)) are supported for the following material data tables:

Material Model	TB,Lab Value
Creep	CREEP
Mass density	DENS
Elasticity	ELASTIC
Extended Drucker-Prager	EDP
Hyperelasticity	HYPER
Nonlinear plasticity	PLASTIC

Material Model	TB,Lab Value
Rate-dependent plasticity	RATE
Thermal expansion	CTE
Thermal properties	THERM

Valid user-defined field variables are UF01 through UF09.

Use your field variables with [initial state loading](#) to define a field of material properties. When you define a field variable as node-based field data over the finite element model (**INISTATE**), the solver reads the initial-state data at the nodes and determines the correct value of the material property necessary for performing the finite element calculations. Initial state loading with user-defined variables defines material models as continuous functions over the models, thus facilitating advanced modeling such as functionally-graded composite materials.

Nonlinear material models sometimes include both linear and nonlinear parameters in the equation.

For example, in the equation $Y = a * x^b$, a is a linear parameter and b is a nonlinear parameter. You might have values of a and b at two states, $UF01 = T1$ and $UF01 = T3$. It may not always be meaningful to interpolate values of b to obtain coefficients at an intermediate field variable value $UF01 = T2$. It is best to keep the values of the nonlinear parameters constant at all states during the [curve-fitting process](#) (p. 259) for these material models.

The following example input shows how to define Young's Modulus as a function of global Y using the elastic material model and the node-based initial state capability:

Example 8.1: Defining Young's Modulus as Function of Global Y

```
TB, ELASTIC, 1
TBFIELD, UF01, 0.0
TBDATA, 1, 1e6, 0.3
TBFIELD, UF01, 1.0
TBDATA, 1, 1e7, 0.3

! Define a UF01 field over the finite element model
INISTATE, SET, NODE, 1
INISTATE, SET, DTYPE, UF01
*GET, NumNodes, node, , num, max
*DO, iI, 1, NumNodes
iExis=NSel(iI)
*IF, iExis, eq, 1, then
yval=NY(iI)
INISTATE, DEFINE, iI, , , yval
*ENDIF
*ENDDO
```

8.2.1. Subroutine for Editing Field Variables

During the solution, you can edit any initialized user-defined field variables via the [UserFld](#) subroutine. Following is a description of the API:

```
! Routine to read current value
!
Call fldgetvar( variable type, curvalue )
Variable type can be any of the user field variables 1-10 and are names such as
FLD_USER_1_TYPE, FLD_USER_2_TYPE, ... FLD_USER_9_TYPE
The current value of the field variable is returned in the double precision
number curvalue
!
```

```

!
! Routine to update current value
!
Call fldsetvar( variable type, curvalue )
Variable type can be any of the user field variables 1-10 and are names such as
FLD_USER_1_TYPE, FLD_USER_2_TYPE, ... FLD_USER_9_TYPE
The current value of the field variable is set to the double precision
number curvalue provided/newly calculated by the user

```

For more information, see [Subroutine userfld \(Update User-Defined Field Variables\)](#) in the *Programmer's Reference*

You can use any appropriate API subroutine in the *Programmer's Reference* to query and update the user-defined field variables.

8.3. Data Processing

Consider a case where isotropic friction (**TB,FRIC**) field data is dependent on both temperature and sliding distance. Assume the following command input:

```

TB,FRIC,1,2, ,ISO
TBFIELD,TEMP,100.0
TBFIELD,SLDI,0.1
TBDATA,1,0.3
TBFIELD,SLDI,0.5
TBDATA,1,0.5
TBFIELD,TEMP,200.0
TBFIELD,SLDI,0.2
TBDATA,1,0.2
TBFIELD,SLDI,0.7
TBDATA,1,0.1

```

Listing the data shows a user-defined 4x2 grid:

```

(FRIC) Table For Material      1

      Data for Isotropic friction

      TEMPERATURE   =  100.00
      SLIDE DIST    =  0.10000

      FRICTION DIR.      FRICTION COEFF.
           1              0.30000

      TEMPERATURE   =  100.00
      SLIDE DIST    =  0.50000

      FRICTION DIR.      FRICTION COEFF.
           1              0.50000

      TEMPERATURE   =  200.00
      SLIDE DIST    =  0.20000

      FRICTION DIR.      FRICTION COEFF.
           1              0.20000

      TEMPERATURE   =  200.00
      SLIDE DIST    =  0.70000

      FRICTION DIR.      FRICTION COEFF.
           1              0.10000

```

A tabular format represents the data in the 4x2 grid as shown:

	Sliding Distance			
Temperature	0.1	0.2	0.5	0.7
100	0.3		0.5	
200		0.2		0.1

When defining tabular data, the first specified field variable forms the rows of the table. The subsequent variables form the columns. In this example, Temperature is the first defined field variable.

In this case, the user defined only four out of a possible eight grid locations.

To populate the interpolation search space, the program fills the missing grid points in each row from left to right. If the first or subsequent grid locations of a row are not defined, the program uses the first defined value within the row to backfill the grid. The program then fills any undefined locations within the grid by linearly interpolating between defined points in each row. If the last value(s) along a row are not defined, the program gives them the last previously defined value within that row.

Therefore, based on the defined field-dependent friction values, the program generates the following grid automatically (where values in *italics* represent those provided by the program):

	Sliding Distance			
Temperature	0.1	0.2	0.5	0.7
100	0.3	<i>0.35</i>	0.5	<i>0.5</i>
200	<i>0.2</i>	0.2	<i>0.14</i>	0.1

8.4. Logarithmic Interpolation and Scaling

Specified material parameters (**TBDATA**) can be interpolated (**TBIN**) in both the linear scale and natural log scale. Logarithmic interpolation is available for material table data specified as a function of any single field variable type.

Supported material data-table types (**TB**) include: **AFDM**, **AHYPER**, **BB**, **CGCR**, **CONC**, **CREEP**, **CTE**, **CZM**, **DMGE**, **DMGI**, **EDP**, **ELASTIC**, **FCLI**, **FLUID**, **FRIC**, **HYPER**, **INTER**, **JROCK**, **MC**, **PELAS**, **PERF**, **PM**, **RATE**, **SDAMP**, **SOIL**, **SWELL**, **THERM**, **WEAR**

Example 8.2: Forcing the Interpolation to a Scale of Your Choice

```
TB,ELASTIC,1
! Interpolate Young's modulus in LOG-LINEAR SCALE
TBIN,SCAL,UF01,1,LOG,LINE
! Interpolate Poisson's Ratio in LOG-LOG SCALE
TBIN,SCAL,UF01,2,LOG,LOG
TBFIELD,UF01,0.0
TBDATA,1,1e6,0.3
TBFIELD,UF01,1.0
TBDATA,1,1e7,0.4
```

For more information, see [Interpolation Algorithms](#) (p. 307).

8.5. Interpolation Algorithms

Several interpolation algorithms are available for use with [supported material data tables](#) (p. 314) (**TB**).

[Simple linear interpolation \(p. 308\)](#) is the default for 1-D and 2-D grid data. To obtain meaningful results for multidimensional non-grid data, it is necessary to use one of the [multidimensional interpolation \(p. 310\)](#) algorithms.

You can select an interpolation algorithm or switch between algorithms (**TBIN**).

The following topics for interpolation algorithms are available:

[8.5.1. Simple Linear Interpolation](#)

[8.5.2. Multidimensional Interpolation](#)

[8.5.3. Evaluating Interpolation Algorithm Results](#)

[8.5.4. Material Model Support for Interpolation](#)

[8.5.5. Reference](#)

8.5.1. Simple Linear Interpolation

Simple linear interpolation can be used for one- and two-dimensional grid data:

[8.5.1.1. Simple Linear Algorithm \(One-Dimensional\)](#)

[8.5.1.2. Simple Linear Algorithm \(Two-Dimensional\)](#)

8.5.1.1. Simple Linear Algorithm (One-Dimensional)

To demonstrate the interpolation of data in a sparsely defined [grid \(p. 306\)](#), consider the results of a two-dimensional interpolation at a temperature of 100 and a sliding distance of 0.40. In this case, the program performs only a one-dimensional interpolation because the defined temperature value (100) lies directly on the defined grid field. For this case, the program obtains a friction coefficient value of 0.45 based on the following calculations:

$$y = y_0 + \alpha(y_1 - y_0)$$

Equation (1)

where

$$\alpha = \frac{x - x_0}{x_1 - x_0}$$

Equation (2)

Substituting the tabular values

$$x = 0.4,$$

$$x_0 = 0.2,$$

$$x_1 = 0.5$$

$$y_0 = 0.35$$

$$y_1 = 0.5$$

Substituting these values into [Equation \(2\) \(p. 308\)](#):

$$\alpha = \frac{0.4 - 0.2}{0.5 - 0.2} = \frac{2}{3}$$

Equation (3)

and solving for the interpolated values using [Equation \(1\) \(p. 308\)](#), we obtain

$$y = .35 + \frac{2}{3}(0.5 - 0.35) = 0.45$$

Equation (4)

8.5.1.2. Simple Linear Algorithm (Two-Dimensional)

Two-dimensional interpolation requires data provided in the proper order. If UF01 and TEMP are the two field variables, for example, input the data in the following format:

	UF01 = 100	UF01 = 200	UF01 = 300	UF01 = 400
TEMP = 0	1	2	3	4
TEMP = 10	5			6
TEMP = 20	7		8	9

If **TBFIELD** for TEMP is issued first, all data points with TEMP held constant and UF01 varying are issued. Then TEMP is modified again and data points with UF01 changing are issued.

Consider the case where a true two-dimensional interpolation is required at a temperature of 180 and a sliding distance of 0.40. The program performs three different linear interpolations to determine the property value within the [grid \(p. 306\)](#). When performing two-dimensional interpolation, the program always interpolates first along the two relevant rows of the grid (Temperature in this case), then between the rows.

In this example, the program performs the first interpolation at a temperature of 100 and a sliding distance of 0.4, yielding the result of 0.45 (as shown in [Equation \(4\) \(p. 308\)](#)).

The program performs the second interpolation for a temperature of 200 and a sliding distance of 0.4. In this case, we find that

$$\begin{aligned} x &= 0.4, & x_0 &= 0.2, & x_1 &= 0.5 \\ y_0 &= 0.2 & y_1 &= 0.14 \end{aligned}$$

Substituting these values into [Equation \(2\) \(p. 308\)](#):

$$\alpha = \frac{0.4-0.2}{0.5-0.2} = \frac{2}{3} \quad \text{Equation (5)}$$

and solving for the interpolated values using [Equation \(1\) \(p. 308\)](#), we obtain

$$y = .2 + \frac{2}{3}(0.14-0.2) = 0.16 \quad \text{Equation (6)}$$

Finally, the program performs a third interpolation between the temperature value of 100 and 200 at a sliding distance of 0.4.

$$\begin{aligned} t &= 180, & t_0 &= 100, & t_1 &= 200 \\ y_0 &= 0.45 & y_1 &= 0.16 \end{aligned}$$

Substituting these values into [Equation \(2\) \(p. 308\)](#):

$$\alpha = \frac{180-100}{200-100} = \frac{4}{5} \quad \text{Equation (7)}$$

and solving for the interpolated values using [Equation \(1\)](#) (p. 308), we obtain

$$y = .45 + \frac{4}{5} (0.16 - 0.45) = 0.218$$

Equation (8)

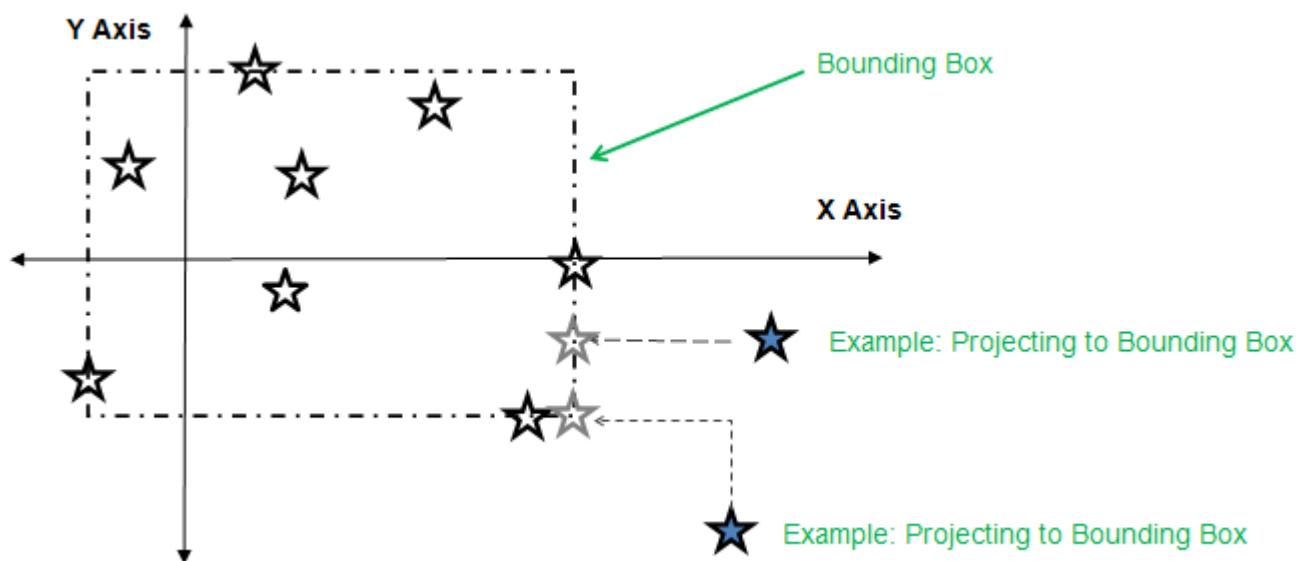
8.5.2. Multidimensional Interpolation

Use the [linear-multivariate](#) (p. 311) algorithm to perform interpolation when two or more field variables are input.

Unlike [simple linear interpolation](#) (p. 308), you can input a random collection of data points where the material property was experimentally evaluated. It is not necessary to input data in any specific grid format (although you can still do so), but you must provide enough data to cover the region where you intend to interpolate.

Because the data is a random sampling, the algorithm creates an n -dimensional rectangular bounding box, and the queries outside of the bounding box are projected to fall on the surface or edge of the bounding box. The algorithm performs the projection by calculating a normal to each axis until it finds a position within the bounding box.

In the following figure, the stars represent the data points input that you provide, and the blue stars represent queries outside of the bounding box:



The figure shows how the data points are projected first in one dimension and then in the second dimension. This method was extrapolated and implemented for multiple dimensions.

Nonlinear interpolation can produce slightly different results from that of linear interpolation.

For material data tables that use **TBPT**-based input (such as **TB,PLASTIC**), stresses are first evaluated at the neighboring strain values provided in the **TBPT** input using the specified multidimensional interpolation algorithm, then linearly interpolated between the neighboring values to maintain the stress strain data C0.

8.5.2.1. Linear-Multivariate Algorithm

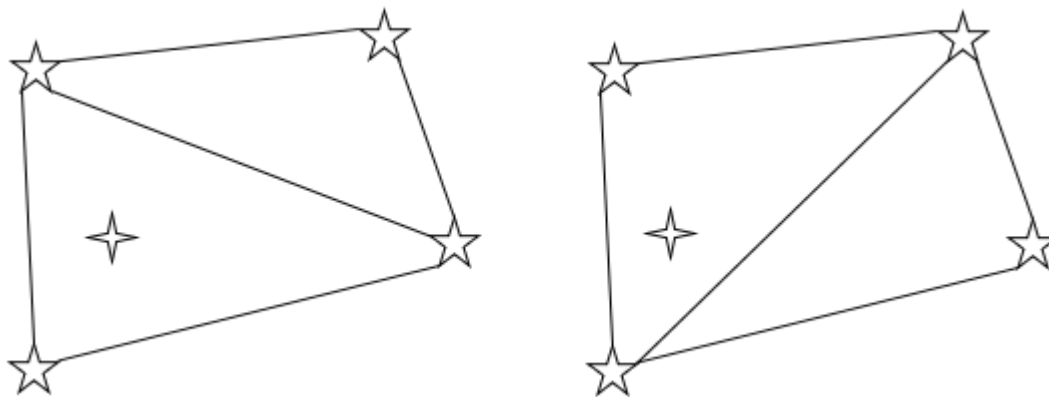
Linear-multivariate is an interpolation algorithm that uses points near the query location to perform the interpolation process. The algorithm generates an n -dimension mesh (simplices) using Delaunay triangulation. It uses that mesh to select an n -dimensional element, and the values at the nodes of the elements, to perform a linear interpolation.

The linear-multivariate algorithm is recommended for parameter spaces of Order 3 and above, or when the data is not a standard 2-D grid. It is highly accurate when given a sufficient distribution of points. (More data points result in greater accuracy.)

When using linear-multivariate interpolation, examine the input and the interpolated values (***GET**) before applying the algorithm to the solution. (See ***GET General Items, Entity = *TBTYPE*** in the [Command Reference](#) and [Evaluating Interpolation Algorithm Results \(p. 313\)](#).) The following example cases indicate the necessity of performing an interpolation study for complex data:

Example 8.3: Linear-Multivariate Interpolation Using Four Points

In some cases, such as a perfect grid, the triangulation may not be unique. In this 2-D example with the given four points, the generated mesh can take either of these forms:



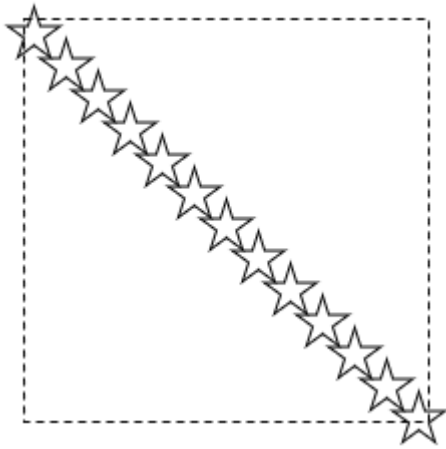
If all four points are not in a plane, the linearly interpolated value is generated either from the bottom three points or from the leftmost three points, depending on how the mesh was generated internally. Enter enough points to ensure that any meshing changes do not affect the results.

Example 8.4: Dimensionality of Data

Linear-multivariate interpolation cannot handle data where the dimensionality of the data points is higher than the dimensionality of the point cloud.

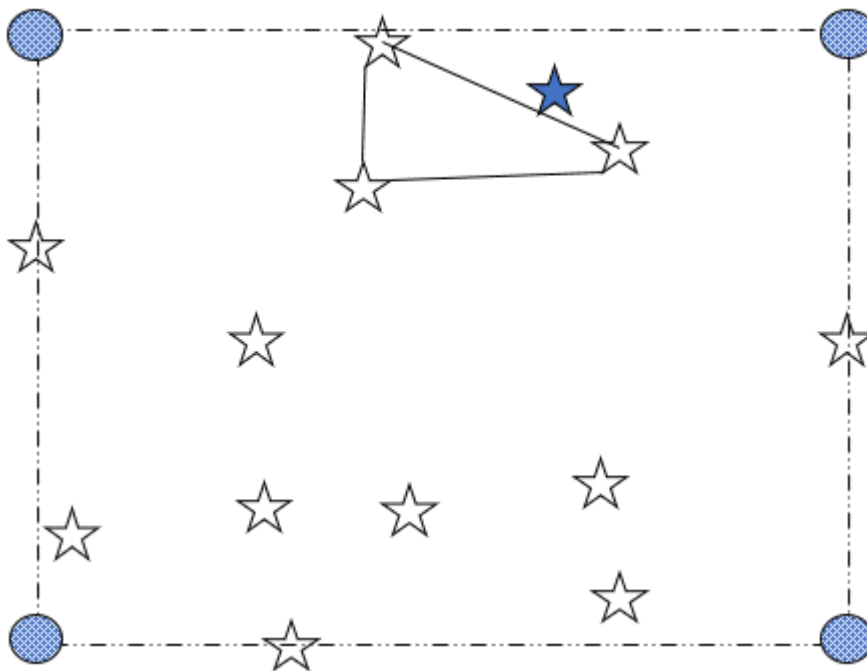
For example, with 2-D data, the algorithm cannot handle cases where all points lie in a single line. You must therefore either transform the data to a single dimension or populate the 2-D data with additional data points to enable the algorithm to create a 2-D mesh.

Likewise, avoid using this algorithm if the data points are 3-D but span only a surface (a dimension less than the dimension of the point) rather than a 3-D volume.



Example 8.5: Extrapolation

Extrapolation situations occurs when the query point (blue star) is specified outside the convex hull of the specified data points but inside the bounding box:



To avoid extrapolation, define the corner points (blue circles) of the bounding box.

If the field variable range differs in the various dimensions, ANSYS, Inc. recommends using normalization (**TBIN,NORM**), enabled by default. For example, use normalization when temperatures range from 100 to 300 and UF01 (such as material fraction) ranges from $1e-5$ to $3e-5$.

For more information about the linear-multivariate algorithm, see [Linear Multivariate Interpolation in the ACP User's Guide](#).

8.5.3. Evaluating Interpolation Algorithm Results

The following example input shows how to input and evaluate the interpolation algorithm via ***GET**:

```
! Example: evaluating interpolation of material data tables
/prep7

! Use a six X six X six grid
xpts=6
ypts=6
zpts=6

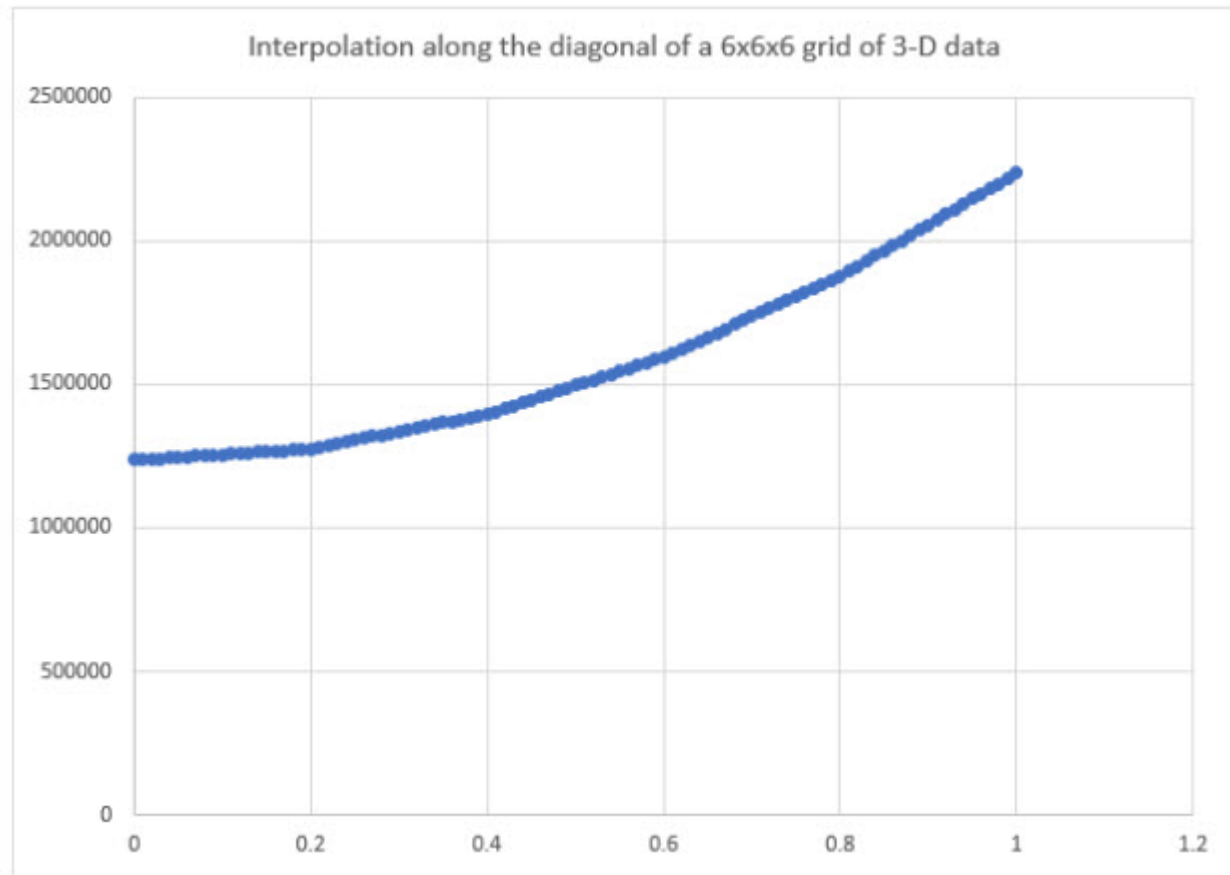
! 3D Grid is unnecessary. The grid below is created for demonstration
! purposes only.

! Setting up a TB,ELAS property with linear-multivariate interpolation algorithm
tb,elas,1
*DO,x,1,xpts,1
*DO,y,1,ypts,1
*DO,z,1,zpts,1
xloc=(x-1)/(xpts-1)
yloc=(y-1)/(ypts-1)
zloc=(z-1)/(zpts-1)
tbfield,xcor,xloc
tbfield,ycor,yloc
tbfield,zcor,zloc
E=(1+xloc**2+yloc**2+zloc**2)*1e6
tbdatt,,E,0.0
*ENDDO
*ENDDO
*ENDDO
tbin,algo,lmul

*dim,ucalc,,101
*dim,zcalc1,,101

! Issue the *GET command to evaluate the value on
! a line across the data
*DO,x,1,101,1
myval=(x-1)/100
ucalc(x)=myval
*get,velas,elas,1,tbev,1,1,3,0.33,0.33,myval
zcalc1(x)=velas
*ENDDO

*stat,ucalc,1,101
*stat,zcalc1,1,101
finish
```

Figure 8.1: Evaluating Interpolation for Quadratic Data with a 6x6x6 Grid

8.5.4. Material Model Support for Interpolation

Simple linear (p. 308) interpolation is supported for [all material data tables](#).

Linear-multivariate (p. 311) interpolation is supported for these material data tables:

Material Model	TB,Lab Value
Coefficient of thermal expansion	CTE
Mass density	DENS
Elasticity	ELASTIC
Hyperelasticity	HYPER
Nonlinear plasticity	PLAS
Porous media	PM

8.5.5. Reference

For further information about multidimensional field-variable interpolation, consult this reference:

1. Amidror, Isaac. "Scattered Data Interpolation Methods for Electronic Imaging Systems: A Survey ." *Journal of Electronic Imaging*. 11:2 (2002): 157-176.

Chapter 9: GUI-Inaccessible Material Properties

The following material properties are not available via the material property menus of the interactive GUI. You can specify them from the command line or input file. Subsequent graphic display and post-processing is still displayed.

TB Command <i>Lab Value</i>	Material Property
AFDM	Acoustic frequency dependent material (p. 240)
AHYPER	Anisotropic hyperelasticity (p. 93)
BB	Bergstrom-Boyce (p. 94)
CDM	Mullins effect (p. 95)
CGCR	Crack growth fracture criterion
CTE	Coefficient of thermal expansion (p. 21)
CZM	Cohesive zone (p. 211)
DENS	Mass density
DMGE	Damage evolution law (p. 221)
DMGI	Damage initiation criteria (p. 220)
EDP	Extended Drucker-Prager (p. 55)
ELASTIC	Elasticity
EXPE	Experimental data (p. 26)
FCLI	Failure criteria material strength limits (p. 218)
FLUID	Fluid (p. 242)
FRIC	Coefficient of friction (p. 204)
GURSON	Gurson pressure-dependent plasticity (p. 64)
INTER	User-defined contact interaction
MPLANE	Microplane (p. 107)
NLISO,,,,POWER	Nonlinear isotropic hardening with power hardening law (p. 43)

TB Command <i>Lab Value</i>	Material Property
PERF	Acoustic perforated medium (p. 233)
PM	Porous media (p. 180)
WEAR	Contact surface wear (p. 214)