

SANDIA REPORT

SAND2017-9759

Unlimited Release

Printed September 11, 2017

Sierra/SolidMechanics 4.46 User's Guide

SIERRA Solid Mechanics Team
Computational Solid Mechanics and Structural Dynamics Department
Engineering Sciences Center

Prepared by
Sandia National Laboratories
Albuquerque, New Mexico 87185 and Livermore, California 94550

Sandia National Laboratories is a multission laboratory managed and operated by National Technology and Engineering Solutions of Sandia, LLC., a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.



Sandia National Laboratories

Issued by Sandia National Laboratories, operated for the United States Department of Energy by National Technology and Engineering Solutions of Sandia, LLC.

NOTICE: This report was prepared as an account of work sponsored by an agency of the United States Government. Neither the United States Government, nor any agency thereof, nor any of their employees, nor any of their contractors, subcontractors, or their employees, make any warranty, express or implied, or assume any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represent that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government, any agency thereof, or any of their contractors or subcontractors. The views and opinions expressed herein do not necessarily state or reflect those of the United States Government, any agency thereof, or any of their contractors.

Printed in the United States of America. This report has been reproduced directly from the best available copy.

Available to DOE and DOE contractors from
U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831

Telephone: (865) 576-8401
Facsimile: (865) 576-5728
E-Mail: reports@adonis.osti.gov
Online ordering: <http://www.osti.gov/bridge>

Available to the public from
U.S. Department of Commerce
National Technical Information Service
5285 Port Royal Rd
Springfield, VA 22161

Telephone: (800) 553-6847
Facsimile: (703) 605-6900
E-Mail: orders@ntis.fedworld.gov
Online ordering: <http://www.ntis.gov/help/ordermethods.asp?loc=7-4-0#online>



Sierra/SolidMechanics 4.46 User's Guide

SIERRA Solid Mechanics Team
Computational Solid Mechanics and Structural Dynamics Department
Engineering Sciences Center
Sandia National Laboratories
Box 5800
Albuquerque, NM 87185-0380

Abstract

Sierra/SolidMechanics (Sierra/SM) is a Lagrangian, three-dimensional code for finite element analysis of solids and structures. It provides capabilities for explicit dynamic, implicit quasistatic and dynamic analyses. The explicit dynamics capabilities allow for the efficient and robust solution of models with extensive contact subjected to large, suddenly applied loads. For implicit problems, Sierra/SM uses a multi-level iterative solver, which enables it to effectively solve problems with large deformations, nonlinear material behavior, and contact. Sierra/SM has a versatile library of continuum and structural elements, and a large library of material models. The code is written for parallel computing environments enabling scalable solutions of extremely large problems for both implicit and explicit analyses. It is built on the SIERRA Framework, which facilitates coupling with other SIERRA mechanics codes. This document describes the functionality and input syntax for Sierra/SM.

Acknowledgments

This document is the result of the collective effort of many individuals. The current core development team responsible for the Sierra/SM codes includes Nathan K. Crane, Gabriel J. de Frias, San Le, David J. Littlewood, Mark T. Merewether, Matthew D. Mosby, Kendall H. Pierson, Julia A. Plews, Vicki L. Porter, Timothy R. Shelton, Jesse D. Thomas, Michael R. Tupek, Michael G. Veilleux, and Patrick G. Xavier. This document is written and maintained by this team.

Outside the core development team, there are many other individuals who have contributed to this manual. Nicole L. Breivik, J. Franklin Dempsey, Jeffery D. Gruda, Kevin N. Long, Kyran D. (Kim) Mish, and Chi S. (David) Lo have provided valuable input from the user community as Product Managers.

Many others have contributed to this document, either directly or indirectly. These include, but are not limited to Manoj K. Bhardwaj, Arthur Brown, James V. Cox, David M. Day, James W. Foulk III, Jason D. Hales, Brian D. Giffin, Steven P. Gomez, Arne S. Gullerud, Daniel C. Hammerand, Martin W. Heinsteins, David M. Hensinger, Michael C. Hillman, Jason T. Ivey, Timothy D. Kostka, J. Richard Koterass, Alex J. Lindblad, Edward Love, Jakob T. Ostien, Rhonda K. Reinert, Nathaniel S. Roehrig, William M. Scherzinger, Gregory D. Sjaardema, Benjamin W. Spencer, Brian L. Stevens, Julia R. Walker, Gerald W. Wellman, and Edouard Yreux.

Sierra/SM 4.46 Release Notes

Following is a list of new features and syntax changes made to Sierra/SM since the 4.40 release.

New features

RBE3 rigid body element constraints are now supported. RBE3's distribute applied forces and moments at a set of “reference” nodes as corresponding forces to a set of “connection” nodes. Mass-weighted distribution of the reference nodal masses to the connection nodes is performed in the computation, which may be optionally deactivated. Refer to Section [7.14.12](#) for details and examples.

Several new output options are available, including

- the internal reaction on a plane within the model.
- the scalar magnitude of any non-scalar global variable, such as the length of a global vector.
- the interpolated value of an output quantity at an arbitrary spatial location in the model.

Consult Sections [9.3.2](#) and [9.3.2.2](#) for additional details.

A new command to set the frequency with which mass is updated is now available. See Section [3.3.2](#) for details.

Element and node IDs are now accessible as variables `element_id` and `node_id`, respectively. These field variables can be used in user output expressions, user subroutines, etc.

The element variables `strain_energy` and `strain_energy_density` are now available for shell and membrane elements.

Contact-specific lofting and thickness may now be included in shell, membrane, and layered shell section definitions.

Beam section visualization objects can now display quantities at integration points. See Section [6.2.7](#) for details.

Several material state variables have been enabled for uncertainty quantification: for the elastic fracture model, `MAX_STRESS` and `CRITICAL_CRACK_OPENING_STRAIN` have been enabled for UQ, and Johnson–Cook and Johnson–Cook Shell models support `YIELD_STRESS`.

Components of multi component global, element, and nodal fields can now be directly accessed in analytic input deck expressions via the bracket `[]` syntax. See Section [2.1.5](#).

The traction boundary condition can now compute traction distributions from an input mesh field.

See Section 7.6.3 for details.

Contact constraints may now be output in Salinas format for preload handoff analyses. See Section 8.10.1 for details.

In cylindrical joints, the parameter `axial_direction` may be specified to define the axial direction of the cylindrical joint. Sierra/SM uses this direction to determine whether the normal direction of the beam element should be reversed. If `axial_direction` is not specified, the default beam normal direction will be used.

The default heartbeat output format now conforms to a column-separated value format. If desired, an option exists to reproduce the previous heartbeat output behavior. See Section 9.5 for details.

Solution Termination using the `WRITE OUTPUT` command now writes results, history, and restart files when the solution termination criteria is reached.

The rebalance command now accepts active or inactive periods as input. See Section 6.8 for details.

A new, specialized initial condition option has been added to aid in defining arbitrary body-fitted coordinate systems. See Section 7.3 for details and a small example.

Input mesh topology is now checked by default. A fatal error will occur if mesh initialization detects any invalid connectivity that is likely to cause problems later in the run. The mesh topology check may be manually disabled. See Section 2.4 for details.

Fixes, performance, and robustness improvements

The field ‘variance’ user output option was previously non-functional and always reported zero. This has been fixed.

Proximity and nodal ID mappings of superelement interface nodes now function in parallel, across mesh decomposition boundaries.

The convention for external force and internal energy in implicit runs has been changed to match explicit behavior. Previously, the reaction load from kinematic boundary conditions was not included in the external force. Similarly, the energy contribution to the system from kinematic boundary conditions was not included in the energy accounting of the model. Now, implicit analyses follow the explicit convention: reaction force is accumulated into external force and kinematic energy is accumulated into total model external energy.

Cylindrical joint radial response is now evaluated in a cylindrical coordinate system. This primarily affects joints with a nonlinear radial gap response. Previously, this gap was evaluated in a Cartesian coordinate system and was thus more “square” than cylindrical. See Section 6.2.15 for details.

Active and inactive periods are now properly respected by the stiffness damping option.

Different element death commands with different numbers of death steps now interact properly.

The solid section `COORDINATE SYSTEM` has been renamed `RVE COORDINATE SYSTEM` to clarify that this command only is used in relation to RVE analysis.

Deprecated features

The element death option `INSERT COHESIVE ZONE` has been deprecated due to robustness issues. This option may be removed entirely in a future release.

Following is a list of new features and syntax changes made to Sierra/SM since the 4.42 release.

A new contact friction model `INACTIVE FRICTION MODEL` is now available. This friction model produces no force and can be used to selectively turn off contact for certain interactions or during certain time periods. See Section [8.5.6](#) for details.

A new appendix was added listing other pre- and post-processing scripts supported by the development team. See Appendix [H](#).

The user output coordinate system transformation operation now supports cylindrical coordinate systems. Additionally the documentation of this option has been greatly expanded. See Section [9.3.2.6](#) for details.

Artificial strains can now access general analytic functions (functions making use of expression variables.) This can be used to define much more general time and space fields for artificial strain.

A new library subroutine was added `aupst_preload_solver`. This subroutine can be used to do a basic predictor corrector iterations in time for unknown preload values. As an example determining the artificial strain required in various bolts to obtain a target clamping force. See Section [11.5.1](#) for details.

`A T AXIS` provided to a beam block now also affects the orientation of the lofted beam contact prism.

The nodal variable `contact_energy` is now available recording work done by contact on a node by node basis.

A new `ABSOLUTE SEARCH TOLERANCE` is available in procedural transfers that allows the user to control the sizes of the search bounding boxes. See Section [12.2](#) for details.

The `L2_PROJECTION` transfer is now implemented for use with the `STRONGLY_OBJECTIVE` form of the `SELECTIVE_DEVIATORIC` and `FULLY_INTEGRATED` hex elements. This capability significantly improves remesh/remap use-cases.

A new nonlinear hourglass formulation (`HOURLASS FORMULATION = HYPERELASTIC`) is available for uniform-gradient hexahedral elements using strongly objective strain incrementation. This hourglass formulation is especially useful when elements are undergoing severe deformation. See Section [6.2.1](#)

The `C0` triangular shell element `FORMULATION = C0_TRI_SHELL` is now available for use in an implicit simulation. See Section [6.2.4](#) for more information about this shell element.

Superelements can now be used in a co-rotational form making them rigid body invariant. Superelements can now be used with a consistent mass matrix. See the capabilities in development manual for details.

It is no longer required to specify `COMPUTE CONTACT VARIABLES = ON` in the input file. Now, Sierra/SM will determine if a contact variable is being requested for output and compute that variable by default at every output time (unless it is an accumulated contact variable). See Section [8.10](#) for more details

Block information involving mass scaling has been added to the log file.

Mass and stiffness damping are now available for use in implicit simulations.

Fixes, performance, and robustness improvements

The user mount element command `ZERO LENGTH R AXIS` now always sets the mount r-axis whether the mount initially has zero length or not.

Improvements to contact have been made that improve conservation of energy during frictional sliding.

A bug was fixed in the Tied MPC's so they can now be applied component-by-component. See Section [7.14.9](#) for more details about this capability.

A bug was fixed in the periodic boundary conditions so they can now be applied component-by-component. See Section [7.4.5](#) for more details about this capability.

Energy reporting has been improved.

Substantial improvements have been made to transfer of multi-integration point elements between Sierra/SM and Sierra/TF (arpeggio) previously integration point locations could have been scrambled during these transfers.

When using layered shells the name of artificially created layer blocks were changed from `block_LAYER` to `block_SIERRA_CREATED_LAYER`. This was done to avoid confusion if user named blocks were inadvertently given a name ending `_LAYER` which confused the output system.

A new user output option was added to calculate the weighted average of a nodal or element variable over a set of nodes or elements.

Two new element fields are available when requested for output `iplocation`, the integration point locations of the element and `centroid` the mass centroid of an element. Previously these fields were available when using certain materials, now they can always be output in any analysis. See Table [9.10](#) for more details.

Restartwrite may now be requested on wall clock time. See Section [9.6.1.1](#) for details.

Following is a list of new features and syntax changes made to Sierra/SM since the 4.44 release.

An additional option was added to the viscous, mass, and velocity damping capability to make the damping forces rigid body motion invariant. See Section [7.14.6.2](#) for details.

Various boundary condition definition options such as `SCALE FACTOR`, `FUNCTION`, and `READ VARIABLE` may now be combined more freely. In such cases, the different options are multiplied together multiplicatively. Additionally, the `READ VARIABLE` command has been added to a num-

ber of BC's that did not support it previously.

If a node has both a MPC constraint and a contact constraint, both constraints will try to be satisfied. Previously in such cases, the MPC constraint would overwrite and invalidate a contact constraint. This new capability is done automatically, but still has some limitations such as not fully handling more complex multi node MPCs.

`CONTROL DAMPED SOLVE` has been implemented to help convergence with difficult implicit quasi-static problems. It is especially useful in contact problems that have intermittent rigid body modes. See Section [4.8](#) for details.

Hex elements may now calculate a `SCALED_JACOBIAN` shape metric that is equivalent to the Cubit 'scaled jacobian'. Additionally, the `ELEMENT_SHAPE` metric computed by hex elements now gives identical results to the Cubit 'shape' metric.

The user output command:

`COMPUTE ELEMENT <target> AS AVERAGE OF ELEMENT <source>` now computes a volume weighted average of the quantity. Previously this command would in some cases compute an arithmetic average. This volumetric average is more consistent with how this command has always been documented.

The default node normal computation used for node/face contact has been changed. The new default angle weighting option is generally more accurate but can be a bit slower than the old area weighting option. See Section [8.9.2](#) for details. Additionally the heuristics used to find node/face interactions at sharp corners has been improved to make it more likely for such corners to find tied contact.

The ability to copy a node, element, or face variable from a results file generated in a previous simulation has been implemented in the `TIME STEP INITIALIZATION` block. An example use case for this is if a user would like to kill elements based off of a temperature field generated in a previous run. See Section [11.2.3.2](#) for details.

The `BEGIN IDENTIFY FRAGMENTS` command now outputs the force and moment of each fragment on a per fragment basis. See Section [9.12](#) for details on this capability.

It is now possible to interpolate a nodal field to an element field in user output. See Section [9.3.2.2](#) for additional details.

The `TIME VARIANT FRICTION MODEL` is now supported in implicit contact runs. See Section [8.5.11](#) for details about this friction model.

A search configuration for `TIED MPC` may now be specified as either model or current coordinates. See Section [7.14.9](#) for more details.

A ParaView timeseries file can now be written via the `TIMESERIES NAME` command in the `RESULTS OUTPUT` block. This is useful for simulations on high numbers of processors with many mesh modifications that output a large number of files. See Section [9.3.1](#) for further details.

When initializing a variable using the `INITIAL CONDITION` block, a time can now be specified at which to read the variable from the separate results file. See Section [7.1.3](#) for further details.

Fixes, performance, and robustness improvements made to Sierra/SM since the 4.44 release.

The explicit Dash contact frictional energy computation has been significantly improved.

The Thouless-Parmigiani cohesive model has been improved for cohesive zones in compression.

Performance of the contact search algorithm has been improved, especially for problems run on more than 1000 MPI ranks.

When applying a gravity boundary condition to a body containing a superelement, the body loads are now applied consistently compared to a standard FEA mesh.

The Selective Deviatoric element now works with `DEATH ON INVERSION`.

Implicit element death was made more consistent with explicit element death. The death criteria is now evaluated at the beginning and the end of each loadstep.

Deprecated features since the 4.44 release.

The `MIDPOINT_INCREMENT` strain incrementation option is no longer recognized by the selective deviatoric element. The selective deviatoric element will now always use the strongly objective strain incrementation which is more accurate.

The ability to specify two DASH-AL contact definitions within a single Adagio region has been deprecated. Please use the time-variant friction model for those use cases.

The computation of `contact_frictional_energy_density` has been fixed to correctly report and preserve frictional energy when the node leaves the contact patch.

The contact node set inclusion convention has been modified to match the contact surface inclusion convention. If a node is potentially in multiple node sets, it is included in only the first set listed in the input deck. This change will generally prevent spurious duplicate interactions at the shared node. It may affect behavior of existing models. See Section [8.4.5](#) for details.

The ten-node composite tetrahedral section documentation has been moved from Capabilities in Development to the User Manual, Section [6.2.16](#).

Contents

1	Introduction	43
1.1	Document Overview	44
1.2	Running The Code	46
1.3	Overall Input Structure	47
1.4	Conventions for Command Descriptions	50
1.4.1	Key Words	50
1.4.2	User-Specified Input	50
1.4.3	Optional Input	51
1.4.4	Default Values	51
1.4.5	Multiple Options for Values	51
1.4.6	Known Issues and Warnings	52
1.4.7	Commands Applicable to Only Explicit or Implicit Analyses	52
1.5	Style Guidelines	54
1.5.1	Comments	54
1.5.2	Continuation Lines	54
1.5.3	Capital and Lower Case Letters	54
1.5.4	Commas and Tabs	54
1.5.5	Blank Spaces	54
1.5.6	General Format of the Command Lines	55
1.5.7	Delimiters	55
1.5.8	Order of Commands	56
1.5.9	Abbreviated END Specifications	56
1.5.10	Indentation	57
1.5.11	Aprepro Text Processing	57
1.5.12	Including Files	57

1.6	Exodus II Database Naming Conventions	58
1.7	Major Scope Definitions for an Input File	59
1.8	Input/Output Files	61
1.9	Obtaining Support	63
1.9.1	Helpful Information to Include when Requesting Support	63
2	General Commands	65
2.1	SIERRA Scope	65
2.1.1	SIERRA Command Block	65
2.1.2	Title	65
2.1.3	Restart Control	66
2.1.3.1	Restart Time	66
2.1.3.2	Automatic Restart	66
2.1.4	User Subroutine Identification	67
2.1.5	XY Functions	67
2.1.6	General Functions	76
2.1.7	Axes, Directions, and Points	79
2.1.8	Orientation	80
2.1.9	Coordinate System Block Command	84
2.1.10	Define Coordinate System Line Command	85
2.2	Procedure and Region	87
2.2.1	Procedure	88
2.2.2	Time Control	88
2.2.2.1	Explicit Dynamic Time Control	88
2.2.2.2	Implicit Time Control	89
2.2.3	Region	89
2.3	Use Finite Element Model	90
2.4	Mesh Quality	90
2.5	Element Distortion Metrics	91
2.6	Activation/Deactivation of Functionality	93
2.7	Error Recovery	93
2.8	Manual Job Control	94
2.9	Solution Termination	96

2.9.1	Solution Termination Examples	97
3	Explicit Dynamic Time Step Control	99
3.1	Procedure Time Control	100
3.1.1	Command Blocks for Time Control and Time Stepping	102
3.1.2	Initial Time Step	103
3.1.3	Time Step Scale Factor	104
3.1.4	Time Step Increase Factor	104
3.1.5	Step Interval	104
3.1.6	Time Control Example	105
3.1.7	Explicit Quasistatic Mode	106
3.1.7.1	Usage Guidelines	107
3.1.7.2	Example Problem	107
3.2	Other Critical Time Step Methods	109
3.2.1	Lanczos Method	110
3.2.1.1	Lanczos Method with Constant Time Steps	111
3.2.1.2	Controls for Lanczos Method	113
3.2.1.3	Scale Factor for Lanczos Method	115
3.2.1.4	Accuracy of Eigenvalue Estimate	115
3.2.1.5	Lanczos Parameters Command Block	116
3.2.2	Power Method	119
3.2.2.1	Power Method with Constant Time Steps	121
3.2.2.2	Controls for Power Method	122
3.2.2.3	Scale Factor for Power Method	122
3.2.2.4	Accuracy of Eigenvalue Estimate	123
3.2.2.5	Power Method Parameters Command Block	123
3.2.3	Node-Based Method	125
3.2.3.1	Node-Based Parameters Command Block	126
3.3	Mass Scaling	128
3.3.1	What is Mass Scaling?	128
3.3.2	Mass Scaling Command Block	128
3.3.3	Node Set Commands	129
3.3.3.1	Mass Scaling Commands	130

3.3.3.2	Additional Commands	130
4	Implicit Solver and Time Step Control	133
4.1	Multilevel Solver	135
4.1.1	Common Multilevel Solver Commands for All Controls	138
4.2	Conjugate Gradient Solver	140
4.2.1	Convergence Commands	142
4.2.2	Preconditioner Commands	145
4.2.3	Line Search Command	148
4.2.4	CG Algorithm Commands	150
4.2.5	Diagnostic Output Commands	152
4.2.6	CG Input Example	152
4.3	Full Tangent Preconditioner	153
4.3.1	Solver Selection Commands	155
4.3.2	Matrix Formation Commands	155
4.3.3	Reset and Iteration Commands	157
4.3.4	Smoothing Iterations	159
4.3.5	Fall-Back Strategy Commands	160
4.3.6	CG with Full Tangent Preconditioner Input Example	161
4.4	FETI Equation Solver	162
4.4.1	Convergence Commands	163
4.4.2	Memory Usage Commands	164
4.4.3	Solver Commands	164
4.4.4	CG with Full Tangent Preconditioner and FETI Input Example	165
4.5	Control Contact	167
4.5.1	Control Contact Commands	168
4.5.1.1	Convergence Commands	168
4.5.1.2	Tips on Setting Control Contact and CG tolerances	170
4.5.1.3	Augmented Lagrange Initialization Commands	170
4.5.1.4	Augmented Lagrange Adaptive Penalty Commands	171
4.5.1.5	Augmented Lagrange Tolerance	172
4.5.1.6	Augmented Lagrange Miscellaneous Options	172
4.5.2	Augmented Lagrange Control Contact	172

4.5.2.1	Augmented Lagrange Control Contact Log File Example	173
4.5.3	Kinematic Control Contact	174
4.5.4	CG with Control Contact Input Example	176
4.6	Control Stiffness	177
4.6.1	Convergence Commands	178
4.7	Control Failure	181
4.7.1	Convergence Command	182
4.7.2	Level Command	182
4.7.3	Other Commands	182
4.7.4	Control Failure Input Example	182
4.8	Control Damped Solve	184
4.8.1	Convergence Command	185
4.8.2	Level Command	186
4.8.3	Control Damped Solve Artificial Spring Commands	186
4.8.4	Other Commands	187
4.8.5	Control Damped Solve Input Example	187
4.9	Predictors	188
4.9.1	Loadstep Predictor	188
4.9.1.1	Predictor Type	188
4.9.1.2	Scale Factor	189
4.9.1.3	Slip Scale Factor	189
4.9.2	Level Predictor	190
4.9.3	Loadstep Predictor Input Example	190
4.10	Implicit Time Step Control	191
4.10.1	Command Blocks for Time Control and Time Stepping	191
4.10.1.1	Time Increment	193
4.10.1.2	Number of Time Steps	193
4.10.1.3	Time Increment Function	193
4.10.2	Adaptive Time Stepping	193
4.10.2.1	Method	194
4.10.2.2	Target Iterations	195
4.10.2.3	Iteration Window	195

4.10.2.4	Cutback Factor	196
4.10.2.5	Growth Factor	196
4.10.2.6	Maximum Failure Cutbacks	196
4.10.2.7	Maximum Multiplier	196
4.10.2.8	Minimum Multiplier	197
4.10.2.9	Reset at New Period	197
4.10.2.10	Active Periods	197
4.10.2.11	User Criteria Global	198
4.10.3	Time Control Example	198
4.11	Implicit Dynamic Time Integration	199
4.11.1	Implicit Dynamics	199
4.11.1.1	Active Periods	199
4.11.1.2	Use HHT Integration	199
4.11.1.3	HHT Parameters	199
4.11.1.4	Implicit Dynamic Timestep Cutback for Contact	200
4.12	Deprecated Commands	200
5	Material Models	203
5.1	General Material Commands	207
5.1.1	Density Command	207
5.1.2	Biot's Coefficient Command	207
5.1.3	General Artificial Strains	207
5.1.4	Thermal Strain Behavior	208
5.1.4.1	Defining Thermal Strains	209
5.1.4.2	Activating Thermal Strains	210
5.1.4.3	Thermal Strain Example	211
5.1.5	Elastic Constants	212
5.1.6	The Isotropic/Kinematic hardening BETA parameter	214
5.1.7	Coordinate Systems	214
5.2	Material Models	217
5.2.1	Elastic Model	217
5.2.2	Thermoelastic Model	219
5.2.3	Neo-Hookean Model	220

5.2.4	Elastic Fracture Model	222
5.2.5	Elastic-Plastic Model	224
5.2.6	Elastic-Plastic Power-Law Hardening Model	228
5.2.7	Ductile Fracture Model	231
5.2.8	Multilinear EP Hardening Model	234
5.2.9	Multilinear EP Hardening Model with Failure	238
5.2.10	Elastic-Plastic Hardening Model with Failure	243
5.2.11	Thermoelastic-Plastic Model	244
5.2.12	Thermoelastic-Plastic Model with Failure	246
5.2.13	Hosford Plasticity Model	248
5.2.14	Hill Plasticity Model	252
5.2.15	Barlat Plasticity Model	258
5.2.16	Johnson–Cook Model	265
5.2.16.1	Command lines	266
5.2.16.2	Warnings and Usage Guidelines	267
5.2.17	BCJ Model	268
5.2.17.1	Warnings and Usage Guidelines	269
5.2.18	BCJ_MEM Model	271
5.2.18.1	Warnings and Usage Guidelines	277
5.2.19	Power Law Creep Model	279
5.2.20	Viscoplastic Model	281
5.2.21	Soil and Crushable Foam Model	286
5.2.22	Foam Plasticity Model	289
5.2.23	Low Density Foam Model	293
5.2.24	Hyperfoam Model	295
5.2.25	Elastic Three-Dimensional Orthotropic Model	297
5.2.26	Elastic Orthotropic Shell Model	301
5.2.27	Elastic Orthotropic Model	303
5.2.28	Elastic Orthotropic Fail Model	307
5.2.28.1	Warnings and Usage Guidelines	312
5.2.29	Wire Mesh Model	315
5.2.30	Orthotropic Crush Model	317

5.2.31	Orthotropic Rate Model	321
5.2.31.1	Warnings and Usage Guidelines	323
5.2.32	Plane Stress Rate Plasticity	324
5.2.32.1	Default Behavior: USER RATE DEPENDENCE = 0	325
5.2.32.2	Alternate Behavior: USER RATE DEPENDENCE = 1	326
5.2.32.3	Other Options	327
5.2.32.4	Warnings and Usage Guidelines	328
5.2.33	Incompressible Solid Model	329
5.2.34	Mooney-Rivlin Model	332
5.2.35	Stiff Elastic	335
5.2.36	Swanson Model	337
5.2.37	Viscoelastic Swanson Model	340
5.2.38	Additional Material Models in Development	345
5.3	Cohesive Zone Material Models	346
5.3.1	Traction Decay	346
5.3.2	Tvergaard Hutchinson	347
5.3.3	Thouless Parmigiani	349
5.3.4	Compliant Joint	351
5.3.5	Functional Interface Traction	352
5.3.6	Alves-Roehl	353
6	Elements	357
6.1	Finite Element Model	358
6.1.1	Identification of Mesh File	361
6.1.2	Alias	362
6.1.3	Omit Block	362
6.1.4	Time Scale Factor	363
6.1.5	Component Separator Character	363
6.1.6	Element Block Parameters	363
6.1.6.1	Material Property	365
6.1.6.2	Include All Blocks	365
6.1.6.3	Remove Block	366
6.1.6.4	Section	366

6.1.6.5	Linear and Quadratic Bulk Viscosity	367
6.1.6.6	Hourglass Control	367
6.1.6.7	Effective Moduli Model	369
6.1.6.8	Element Numerical Formulation	370
6.1.6.9	Activation/Deactivation of Element Blocks by Time	371
6.1.7	Element Block Default Parameters	372
6.2	Element Sections	375
6.2.1	Solid Section	378
6.2.1.1	Node Based Tetrahedra	380
6.2.1.2	Q1P0 Formulation	382
6.2.2	Cohesive Section	384
6.2.3	Localization Section	384
6.2.3.1	Kinematics	384
6.2.3.2	Command summary	385
6.2.3.3	Usage guidelines	387
6.2.4	Shell Section	388
6.2.4.1	Orthotropic Properties	395
6.2.5	Layered Shell Section	396
6.2.6	Membrane Section	398
6.2.7	Beam Section	401
6.2.7.1	Reference Axis Offsets	404
6.2.7.2	Section Integration	406
6.2.7.3	Beam Element Accuracy	410
6.2.8	Truss Section	411
6.2.9	Spring Section	412
6.2.10	Damper Section	414
6.2.11	Point Mass Section	414
6.2.12	Particle Section	417
6.2.13	Peridynamics Section	421
6.2.14	User Mount Section	424
6.2.14.1	User Mount API	426
6.2.14.2	Coordinate Frame	426

6.2.14.3	Mount Output Variables	427
6.2.14.4	Mount Sizing Routine	427
6.2.14.5	Mount Initialization Routine	428
6.2.14.6	Mount Execution Routine	430
6.2.14.7	Example	433
6.2.15	Cylindrical Joint	437
6.2.16	Composite Tet	440
6.3	Element-like Functionality	442
6.3.1	Rigid Body	442
6.3.1.1	Multiple Rigid Bodies from a Single Block	446
6.3.2	Output	447
6.4	Mass Property Calculations	448
6.4.1	Block Set Commands	448
6.4.2	Structure Command	449
6.5	Element Death	450
6.5.1	Block Set Commands	452
6.5.2	Criterion Commands	453
6.5.2.1	Nodal Variable Death Criterion	453
6.5.2.2	Element Variable Death Criterion	454
6.5.2.3	Global Death Criterion	455
6.5.2.4	Always Death Criterion	456
6.5.2.5	Material Death Criterion	456
6.5.2.6	Subroutine Death Criterion	457
6.5.3	Evaluation Commands	457
6.5.4	Miscellaneous Option Commands	458
6.5.4.1	Summary Output Commands	458
6.5.4.2	Death on Inversion	458
6.5.4.3	Death on Ill-defined Contact	459
6.5.4.4	Death Steps	459
6.5.4.5	Degenerate Mesh Repair	460
6.5.4.6	Aggressive Contact Cleanup	460
6.5.4.7	Death Method	460

6.5.4.8	Death on Proximity	461
6.5.4.9	Particle Conversion	462
6.5.4.10	Active Periods	463
6.5.5	Cohesive Zone Setup Commands	463
6.5.6	Example	464
6.5.7	Element Death Visualization	464
6.6	Particle Embedding	466
6.7	Explicitly Computing Derived Quantities	467
6.8	Mesh Rebalancing	468
6.8.1	Rebalance	468
6.8.1.1	Rebalance Command Lines	469
6.8.1.2	Zoltan Command Line	470
6.8.2	Zoltan Parameters	470
6.9	Remeshing	472
6.9.1	Remeshing Commands	473
6.9.2	Remesh Block Set	475
6.9.3	Adaptive Refinement	476
6.9.3.1	Adaptive Refinement Control Commands	476
6.9.3.2	Tool Mesh Entity Commands	477
6.9.3.3	Activation Commands	478
6.9.4	Known Issues	478
7	Boundary Conditions and Initial Conditions	481
7.1	General Boundary Condition Concepts	482
7.1.1	Mesh-Entity Assignment Commands	482
7.1.2	Methods for Specifying Boundary Conditions	484
7.1.3	External Mesh Database Commands	485
7.2	Initial Mesh Modification	488
7.3	Initial Variable Assignment	489
7.3.1	Mesh-Entity Set Commands	490
7.3.2	Variable Identification Commands	491
7.3.3	Specification Command	491
7.3.4	Weibull Probability Distribution Commands	492

7.3.5	External Mesh Database Commands	493
7.3.6	User Subroutine Commands	494
7.3.7	Distance Function Commands	495
7.3.8	Additional Command	495
7.3.9	Specialized Initialization Methods	495
7.3.10	Examples	495
7.4	Kinematic Boundary Conditions	497
7.4.1	Fixed Displacement	497
7.4.1.1	Node Set Commands	498
7.4.1.2	Specification Commands	498
7.4.1.3	Additional Commands	498
7.4.2	Prescribed Displacement	499
7.4.2.1	Node Set Commands	500
7.4.2.2	Specification Commands	501
7.4.2.3	User Subroutine Commands	502
7.4.2.4	External Mesh Database Commands	503
7.4.2.5	Additional Commands	503
7.4.3	Prescribed Velocity	503
7.4.3.1	Node Set Commands	505
7.4.3.2	Specification Commands	505
7.4.3.3	User Subroutine Commands	506
7.4.3.4	External Mesh Database Commands	507
7.4.3.5	Additional Commands	507
7.4.4	Prescribed Acceleration	508
7.4.4.1	Node Set Commands	509
7.4.4.2	Specification Commands	510
7.4.4.3	User Subroutine Commands	510
7.4.4.4	External Mesh Database Commands	511
7.4.4.5	Additional Commands	511
7.4.5	Periodic Boundary Condition	512
7.4.6	Fixed Rotation	514
7.4.6.1	Node Set Commands	515

7.4.6.2	Specification Commands	515
7.4.6.3	Additional Commands	515
7.4.7	Prescribed Rotation	516
7.4.7.1	Node Set Commands	518
7.4.7.2	Specification Commands	518
7.4.7.3	User Subroutine Commands	519
7.4.7.4	External Mesh Database Commands	519
7.4.7.5	Additional Commands	520
7.4.8	Prescribed Rotational Velocity	520
7.4.8.1	Node Set Commands	522
7.4.8.2	Specification Commands	522
7.4.8.3	User Subroutine Commands	523
7.4.8.4	External Mesh Database Commands	524
7.4.8.5	Additional Commands	524
7.4.9	Reference Axis Rotation	524
7.4.9.1	Block Command	526
7.4.9.2	Specification Commands	526
7.4.9.3	Rotation Commands	527
7.4.9.4	Torque Command	527
7.4.9.5	Additional Commands	527
7.4.10	Subroutine Usage for Kinematic Boundary Conditions	528
7.5	Initial Velocity Conditions	529
7.5.1	Node Set Commands	530
7.5.2	Direction Specification Commands	530
7.5.3	Angular Velocity Specification Commands	531
7.5.4	User Subroutine Commands	532
7.6	Force Boundary Conditions	533
7.6.1	Pressure	533
7.6.1.1	Surface Set Commands	535
7.6.1.2	Specification Commands	535
7.6.1.3	User Subroutine Commands	536
7.6.1.4	External Pressure Sources	537

7.6.1.5	Output Command	538
7.6.1.6	Additional Commands	539
7.6.1.7	Surface Integration Schemes	539
7.6.1.8	Example	539
7.6.2	Traction	541
7.6.2.1	Surface Set Commands	542
7.6.2.2	Traction Specification Commands	542
7.6.2.3	User Subroutine Commands	543
7.6.3	External Mesh Database Commands	544
7.6.3.1	Additional Commands	544
7.6.3.2	Surface Integration Schemes	545
7.6.3.3	Example	545
7.6.4	Prescribed Force	545
7.6.4.1	Node Set Commands	547
7.6.4.2	Specification Commands	547
7.6.4.3	User Subroutine Commands	548
7.6.4.4	Additional Commands	549
7.6.4.5	Example	549
7.6.5	Prescribed Moment	550
7.6.5.1	Node Set Commands	551
7.6.5.2	Specification Commands	552
7.6.5.3	User Subroutine Commands	552
7.6.5.4	Additional Commands	553
7.7	Distributed Force and Moment	554
7.7.0.5	Node Set Commands	555
7.7.0.6	Load Definition Commands	555
7.7.0.7	Miscellaneous Commands	555
7.7.0.8	Restrictions and Assumptions	555
7.7.0.9	Example: Applying Torque to Solid Bar	556
7.8	Gravity	558
7.9	Centripetal Force	560
7.10	Prescribed Temperature	561

7.10.1	Block Set Commands	562
7.10.2	Specification Command	562
7.10.3	User Subroutine Commands	563
7.10.4	External Mesh Database Commands	563
7.10.5	Coupled Analysis Commands	564
7.10.6	Additional Commands	564
7.11	Pore Pressure	566
7.11.1	Block Set Commands	567
7.11.2	Specification Command	567
7.11.3	User Subroutine Commands	567
7.11.4	External Mesh Database Commands	568
7.11.5	Coupled Analysis Commands	568
7.11.6	Additional Commands	569
7.12	Fluid Pressure	570
7.12.1	Surface Set Commands	571
7.12.2	Specification Commands	571
7.12.3	Additional Commands	572
7.13	Inertia Relief	573
7.13.1	Node Set Commands	573
7.13.2	Component Command	574
7.13.3	Additional Commands	574
7.13.4	General Notes	574
7.13.5	Example	575
7.14	Specialized Boundary Conditions	576
7.14.1	Cavity Expansion	576
7.14.2	Silent Boundary	578
7.14.3	Spot Weld	578
7.14.3.1	Spot Weld Limitations and Usage Guidelines	583
7.14.3.2	Example	584
7.14.4	Line Weld	585
7.14.5	Shell Solid Joint	587
7.14.5.1	Shell Solid Joint in Implicit Computations	588

7.14.5.2	Restrictions and Assumptions	589
7.14.6	Viscous Damping	590
7.14.6.1	Block Set Commands	591
7.14.6.2	Viscous Damping Coefficient	591
7.14.6.3	Additional Command	592
7.14.7	Artificial Strain	592
7.14.7.1	Restrictions and Usage Guidelines	594
7.14.8	Volume Repulsion Old	594
7.14.8.1	Block Set	595
7.14.9	Tied Multi-Point Constraints	595
7.14.9.1	Resolve Multiple MPCs	597
7.14.9.2	Remove Gaps and Overlaps	598
7.14.10	Master Slave Multi-Point Constraints	598
7.14.10.1	Resolve Multiple MPCs	600
7.14.11	Submodel	600
7.14.12	RBE3	601
7.14.12.1	Sample RBE3 Usage	602
8	Contact	605
8.1	Overview of Contact Constraints	605
8.2	Overview of Contact Enforcement	607
8.3	Contact Definition Block	608
8.4	Defining the Contact Surfaces	611
8.4.1	Contact Surface Command Line	611
8.4.2	Contact Node Set Command Line	612
8.4.3	Contact Surface Command Block	612
8.4.3.1	Surface Subsetting	613
8.4.3.2	Known Issues	613
8.4.4	Skin All Blocks Command	614
8.4.5	Faces and Nodes in Multiple Contact Surfaces	614
8.4.6	Element Death, Remeshing, and Surface Updates	616
8.4.7	Block Skinning and Surface Definition Examples	616
8.4.7.1	Simple Block Skinning	617

8.4.7.2	Block Skinning With Multiple Element Blocks	618
8.4.7.3	More Complex Block Skinning Cases	619
8.4.7.4	Combining Block Skinning, Side Sets, and Element Death	620
8.4.7.5	Equivalenced Solid and Shell Meshes	621
8.5	Friction Models	623
8.5.1	Frictionless Model	623
8.5.2	Constant Friction Model	623
8.5.3	Velocity Dependent Coulomb Friction Model	623
8.5.4	Variable Coefficient Coulomb Model	624
8.5.5	Tied Model	625
8.5.6	Inactive Friction Model	625
8.5.7	Dynamic Tied Model	625
8.5.8	Glued Model	626
8.5.9	Cohesive Zone Model	626
8.5.10	Hybrid Model	629
8.5.11	Time Variant Model	630
8.5.12	Nodal Field Dependent Model	631
8.6	Definition of Interactions	634
8.6.1	Interaction Defaults Block	634
8.6.1.1	General Contact and Self-Contact	635
8.6.1.2	Specific Surfaces Identification	635
8.6.1.3	Friction Model	636
8.6.1.4	Interaction Behavior	636
8.6.1.5	Constraint Formulation	636
8.6.1.6	Augmented Lagrange Penalty	637
8.6.2	Specific Interactions	637
8.6.2.1	Surface Identification	638
8.6.2.2	Interaction Behavior	640
8.6.2.3	Constraint Formulation	640
8.6.2.4	Friction Model	640
8.6.2.5	Interface Material	640
8.6.2.6	Tolerances	641

8.6.2.7	Augmented Lagrange Penalty	641
8.6.2.8	Pushback Factor	641
8.6.2.9	Tension Release	642
8.6.2.10	Tension Release Function	642
8.6.3	Interaction Behavior for Particle Element Blocks	642
8.6.4	Face/Face vs. Node/Face constraints	643
8.7	Active Periods	645
8.8	Contact with Analytic Objects	646
8.8.1	Kinematic BC's with Analytic Objects	646
8.9	Contact Algorithm Options	647
8.9.1	Contact Library	647
8.9.1.1	How Dash and ACME differ	647
8.9.1.2	Current Dash Usage Guidelines	647
8.9.2	Dash Specific Options	648
8.9.2.1	Node Normal Smoothing	650
8.9.3	Enforcement	650
8.9.4	Remove Initial Overlap	651
8.9.4.1	Overlap Removal Diagnostics	652
8.9.4.2	Overlap Removal Commands	653
8.9.4.3	Remove Initial Overlap Usage Guidelines	654
8.9.5	Lofted Surface Options	655
8.9.5.1	Lofted Beams	655
8.9.5.2	Lofted Particles	656
8.9.5.3	Lofted Shells	657
8.9.5.4	Coincident Face Options	658
8.9.6	Search Options	659
8.9.6.1	Search Tolerances	659
8.9.7	Enforcement Options	660
8.10	Contact-Specific Outputs	663
8.10.1	Contact Debugging	663
8.11	Log File Contact Information	665
8.12	Examples	666

8.12.0.1	Example 1	666
8.12.0.2	Example 2	667
8.13	Common Contact Algorithmic Issues	670
8.14	ACME Specific Commands	671
8.14.1	ACME Specific Friction Models	671
8.14.1.1	Spring Weld Model	671
8.14.1.2	Surface Weld Model	672
8.14.1.3	Area Weld Model	673
8.14.1.4	Adhesion Model	673
8.14.1.5	Junction Model	674
8.14.1.6	Threaded Model	675
8.14.1.7	PV_Dependent Model	676
8.14.1.8	User Subroutine Friction Models	677
8.14.2	Kinematic Partition	678
8.14.3	ACME automated search tolerances	680
8.14.4	Frictional behavior without Friction Model	681
8.14.5	Shell Overlap Removal Iterations	681
8.14.6	Surface Normal Smoothing	682
8.14.7	Angle for Multiple Interactions	683
8.14.8	ACME Specific Lofted Geometry Options	684
8.14.9	ACME Specific Debugging Options	685
8.15	Analytic Rigid Surfaces	686
8.16	Deprecated Options	687
9	Output	689
9.1	Syntax for Requesting Variables	690
9.1.1	Example 1	690
9.1.2	Example 2	691
9.1.3	Other command blocks	692
9.1.4	Rigid Body Variables	692
9.2	General Output Database Commands	694
9.3	Results Output	696
9.3.1	Exodus Results Output File	696

9.3.1.1	Output Nodal Variables	698
9.3.1.2	Output Node Set or Surface Variables	699
9.3.1.3	Output Face Variables	701
9.3.1.4	Output Element Variables	703
9.3.1.5	Subsetting of Output Mesh	705
9.3.1.6	Output Mesh Selection	706
9.3.1.7	Component Separator Character	706
9.3.1.8	Output Global Variables	706
9.3.1.9	Specifying Time Steps for Output	707
9.3.1.10	Synchronize Output	710
9.3.1.11	Use Output Scheduler	710
9.3.1.12	Write Results If System Error Encountered	710
9.3.2	User-Defined Output	711
9.3.2.1	Mesh-Entity Set Commands	714
9.3.2.2	Compute Result Commands	715
9.3.2.3	User Subroutine Commands	721
9.3.2.4	Copy Command	722
9.3.2.5	Extrapolate Command	722
9.3.2.6	Variable Transformation Command	723
9.3.2.7	Data Filtering Commands	725
9.3.2.8	Compute at Every Step Command	726
9.3.2.9	Additional Commands	726
9.4	History Output	728
9.4.1	Output Variables	729
9.4.1.1	Global Output Variables	729
9.4.1.2	Mesh Entity Output Variables	730
9.4.1.3	Nearest Point Output Variables	731
9.4.2	Outputting History Data on a Node Set	732
9.4.2.1	Specifying Time Steps for Output	732
9.4.3	Synchronize Output	732
9.4.4	Use Output Scheduler	733
9.4.5	Write History If System Error Encountered	733

9.5	Heartbeat Output	734
9.5.1	Output Variables	736
9.5.1.1	Global Output Variables	736
9.5.1.2	Mesh Entity Output Variables	737
9.5.1.3	Nearest Point Output Variables	738
9.5.2	Outputting Heartbeat Data on a Node Set	739
9.5.3	Specifying Time Steps for Output	739
9.5.4	Synchronize Output	739
9.5.5	Use Output Scheduler	739
9.5.6	Write Heartbeat On Signal	740
9.5.7	Heartbeat Output Formatting Commands	740
9.5.7.1	CTH SpyHis output format	741
9.5.7.2	Specify floating point precision	741
9.5.7.3	Specify Labeling of Heartbeat Data	741
9.5.7.4	Specify Existence of Legend for Heartbeat Data	742
9.5.7.5	Specify format of timestamp	742
9.5.8	Monitor Output Events	743
9.6	Restart Data	745
9.6.1	Restart Options	747
9.6.1.1	What Times to Write	747
9.6.1.2	Automatic Read and Write of Restart Files	748
9.6.1.3	User-Controlled Read and Write of Restart Files	750
9.6.1.4	Overwriting Restart Files	752
9.6.1.5	Recovering from a Corrupted Restart	753
9.6.2	Component Separator Character	753
9.6.3	Specifying Time Steps for Output	753
9.6.4	Reducing Restart File Size	753
9.6.4.1	Overlay Count	754
9.6.4.2	Cycle Count	754
9.6.4.3	Example	754
9.6.5	Synchronize Output	755
9.6.6	Use Output Scheduler	755

9.6.7	Write Restart If System Error Encountered	755
9.6.8	Restarting an Implicit/Explicit Analysis	756
9.7	Output Scheduler	757
9.7.1	Output Scheduler Command Block	757
9.7.1.1	Specifying Time Steps for Output	757
9.7.2	Example of Using the Output Scheduler	758
9.8	Variable Interpolation	759
9.8.1	Variable Interpolation Examples	761
9.9	Global Output Options	763
9.10	Variables	764
9.10.1	Global, Nodal, Face, and Element Variables	764
9.10.2	Variables for Material Models	781
9.11	Solution Verification	782
9.12	Identify Fragments	784
9.13	Detect Breach	785
10	Peridynamics	787
10.1	Peridynamics	787
10.1.1	Overview	787
10.1.2	State-Based Linear Peridynamic Solid Material Model	788
10.1.3	Bond-Based Microelastic Peridynamic Material Model	789
10.1.4	Bond-Based Microplastic Peridynamic Material Model	789
10.1.5	Interface to Classical Material Models	789
10.1.6	Peridynamic Influence Function	790
10.1.7	Modeling Fracture	791
10.1.8	Peridynamics and Contact	792
10.1.9	Usage Guidelines	792
11	User Subroutines	795
11.1	Programming	799
11.1.1	Subroutine Interface	799
11.1.2	Query Functions	800
11.1.2.1	Parameter Query	800

11.1.2.2	Function Data Query	805
11.1.2.3	Time Query	805
11.1.2.4	Field Variables	806
11.1.2.5	Global Variables	815
11.1.2.6	Topology Extraction	818
11.1.3	Miscellaneous Query Functions	823
11.2	Command File	826
11.2.1	Subroutine Identification	826
11.2.2	User Subroutine Command Lines	826
11.2.2.1	Type	826
11.2.2.2	Debugging	827
11.2.2.3	Parameters	827
11.2.3	Time Step Initialization	828
11.2.3.1	Mesh-Entity Set Commands	829
11.2.3.2	Read Mesh Variable Commands	830
11.2.3.3	User Subroutine Commands	830
11.2.3.4	Additional Command	831
11.2.4	User Variables	831
11.3	Compilation and Execution	834
11.4	Examples	835
11.4.1	Pressure as a Function of Space and Time	835
11.4.2	Error Between a Computed and an Analytic Solution	838
11.4.3	Transform Output Stresses to a Cylindrical Coordinate System	842
11.5	User Subroutine Library	848
11.5.1	aupst_preload_solver	848
11.5.2	aupst_torque_calc	853
11.5.3	aupst_trans_rot_velocity	854
11.5.4	aupst_cyl_transform	855
11.5.5	aupst_rec_transform	857
11.5.6	copy_data	858
11.5.7	trace	859
11.5.8	aupst_cavity_volume	860

12 Transfers	863
12.1 SIERRA Transfers	863
12.2 Inter-procedural Transfers	864
12.2.1 Copying Data with Inter-procedural Transfers	865
12.2.2 Selecting Variables for Transfer	865
12.2.3 Additional Transfer Options	866
12.2.4 Interpolating Data with Interpolation Transfers	866
12.2.4.1 Nodal variable interpolation transfer	867
12.2.4.2 Element variable interpolation transfer	867
12.2.4.3 Element variable L_2 projection transfer	868
12.3 Automated Implicit/Explicit Handoff	868
12.3.1 Known Issues and Limitations	870
A Explicit Dynamic Example Problem	871
B Implicit Quasistatic Example Problem	881
C Consistent Units	889
D Constraint Enforcement Hierarchy	891
E Improving Explicit Dynamics Performance	893
F Troubleshooting Guide for Implicit Convergence	897
G User-Defined Model Initialization	901
H Other Scripts Maintained by Development Team	905
I Known Issues	907
I.1 Epic Material Model Issues	908
I.2 SPH Particle Method Time Step Stability Issues	909
I.3 Particle Creation Issues	909
I.4 Bad Element Shapes Yielding Invalid Contact Geometry	909
Index	911

List of Figures

1.1	Input/output files	61
2.1	Piecewise linear and piecewise constant functions	70
2.2	Example Displacement Map	77
2.3	Adjacent shell elements with non-aligned local coordinate systems	81
2.4	Rectangular coordinate system	82
2.5	Z-Rectangular coordinate system.	82
2.6	Cylindrical coordinate system.	83
2.7	Spherical coordinate system.	83
2.8	Rotation about 1	84
2.9	Examples of elements with varying nodal Jacobians	91
4.1	Reaching convergence with controls and the multilevel solver.	136
4.2	Contact configuration at the beginning of the contact update.	175
4.3	Contact gap removal (after contact search).	175
4.4	Contact slip calculations.	176
5.1	Isotropic Hardening, Beta = 1.0	215
5.2	Kinematic Hardening, Beta = 0.0	216
5.3	Mixed Hardening, Beta = 0.5	216
5.4	Elastic-Plastic Theory: Example yield surface, f	225
5.5	EP Power Law Hardening Theory: Example Response	229
5.6	Multilinear EP Theory: Example Response	236
5.7	Hosford Plasticity Theory: Hosford yield surface with $a = 4, 8, 100$	250
5.8	Hill Plasticity Theory: Hill yield surface	254
5.9	Barlat Plasticity Theory: Barlat yield surface	262
5.10	Orthotropic Crush Theory: Example Crush Curve	319

6.1	Association between command lines and command block.	367
6.2	The reference B_0 , intermediate B_I , and current configuration B of the body.	385
6.3	Location of geometric plane of shell for various lofting factors.	392
6.4	Local rst coordinate system for a shell element.	393
6.5	Rotation of 30 degrees about the 1-axis (X' -axis).	394
6.6	Definition of shell RST coordinate system	396
6.7	Layered Shell Example	398
6.8	Beam ROD Cross Section	402
6.9	Beam TUBE Cross Section	403
6.10	Beam BAR Cross Section	403
6.11	Beam BOX Cross Section	404
6.12	Beam I Cross Section	404
6.13	Beam L Cross Section	405
6.14	Beam C1 Cross Section	405
6.15	Beam C2 Cross Section	406
6.16	Beam T Cross Section	406
6.17	Beam T1 Cross Section	407
6.18	Beam Z Cross Section	407
6.19	Beam HAT Cross Section	408
6.20	Beam I2 Cross Section	408
6.21	Locations of REFERENCE AXIS command options	409
6.22	User Mount Coordinate System	427
6.23	Cylindrical Joint Axial Rotation	438
6.24	Cylindrical Joint Axial Translation	438
6.25	Cylindrical Joint Radial Rotation	439
6.26	Cylindrical Joint Radial Translation	439
6.27	Examples of how an element is converted and when it is not converted.	463
7.1	Result of a unit pressure load	540
7.2	Result of a unit traction load	546
7.3	Result of a unit prescribed force	550
7.4	Distributed Z Force Load Distribution	556
7.5	Distributed Z Moment Load Distribution	556

7.6	Distributed Load Example	557
7.7	Fluid Pressure Setup	571
7.8	Spot weld force–displacement curves.	580
7.9	Sign convention for spot weld normal displacements.	580
7.10	Spot weld normal and tangent.	581
7.11	Sign convention for spot weld normal displacements with <code>IGNORE INITIAL OFFSET = YES.</code>	583
7.12	Search tolerances for shell-solid joints. (a) input; (b) automatic	588
7.13	RBE3 test positions.	603
8.1	Two blocks in contact.	606
8.2	Faces in Multiple Contact Surfaces: Example 1	615
8.3	Faces in Multiple Contact Surfaces: Example 2	617
8.4	Simple block skinning example	618
8.5	Block skinning with multiple blocks example	618
8.6	Block skinning advanced example	619
8.7	Contact surface definition examples	620
8.8	Hexahedral and shell contact surface definitions	621
8.9	Bricks sharing a cohesive zone and being pulled apart.	627
8.10	Displacement X vs. time for node 8	628
8.11	Cohesive force in x vs. time for node 8	629
8.12	Example usage of the hybrid friction model.	630
8.13	Node/Face Constraint example	644
8.14	Face/Face Constraint example	644
8.15	Node Normal Computation Options	650
8.16	Effect of overlap removal on model geometry.	652
8.17	Lofted beam geometry	656
8.18	Lofted particle geometry	657
8.19	Lofted shell geometry	658
8.20	Example lofted geometries produced by shell lofting	658
8.21	Illustration of normal and tangential tolerances	660
8.22	Problem with two blocks coming into contact	666
8.23	Problem with three blocks coming into contact.	668

8.24	State of bodies after contact and deformation.	669
8.25	Deformed contact geometry.	669
8.26	Illustration of kinematic partition values	680
8.27	Illustrations of multiple interactions at a node.	683
9.1	Default Cylindrical Transformation	724
9.2	Cylindrical System Transformation of Coordinates	724
9.3	Variable interpolation options.	762
9.4	Breach detection between surfaces 1 and 2. Dead elements in red.	785
10.1	Default influence function for peridynamics.	791
11.1	Overview of components required to implement functionality.	798
11.2	Example of Preload Iteration	851
11.3	Anatomy of the Preload Iteration Cycle	852
A.1	Mesh for example problem.	871
A.2	Mesh with blue and green surfaces removed.	872
B.1	Eraser schematic	881
B.2	Complete eraser mesh	882

List of Tables

4.1	Log File Convergence Markings	146
5.1	State Variables for ELASTIC FRACTURE Model	223
5.2	State Variables for ELASTIC PLASTIC Model	227
5.3	State Variables for ELASTIC PLASTIC Model for Shells	227
5.4	State Variables for EP POWER HARD Model	230
5.5	State Variables for EP POWER HARD Model for Shells	230
5.6	State Variables for DUCTILE FRACTURE Model	233
5.7	State Variables for MULTILINEAR EP Model	237
5.8	State Variables for MULTILINEAR EP Model for Shells	237
5.9	State Variables for ML EP FAIL Model	241
5.10	State Variables for ML EP FAIL Model for Shells	242
5.11	State Variables for HOSFORD PLASTICITY Model (Section 5.2.13)	251
5.12	State Variables for HILL PLASTICITY Model (Section 5.2.14)	257
5.13	State Variables for BARLAT PLASTICITY Model (Section 5.2.15)	264
5.14	State Variables for JOHNSON COOK Model	267
5.15	State Variables for BCJ Model	270
5.16	State Variables for BCJ_MEM Model	277
5.17	Additional State Variables for BCJ_MEM Model When Modeling Recrystallization	278
5.18	State Variables for POWER LAW CREEP Model	280
5.19	State Variables for VISCOPLASTIC Model (Section 5.2.20)	285
5.20	State Variables for SOIL FOAM Model	288
5.21	State Variables for FOAM PLASTICITY Model	292
5.22	State Variables for LOW DENSITY FOAM Model	294
5.23	State Variables for ELASTIC ORTHOTROPIC Model (1)	305
5.24	State Variables for ELASTIC ORTHOTROPIC Model (2)	306

5.25	Additional State Variables for ELASTIC ORTHOTROPIC FAIL Model	313
5.26	State Variables for WIRE MESH Model	316
5.27	State Variables for ORTHOTROPIC CRUSH Model	320
5.28	State Variables for ORTHOTROPIC RATE Model	323
5.29	State Variables for ELASTIC PLASTIC Model	328
5.30	State Variables for MOONEY RIVLIN Model	334
5.31	State Variables for SWANSON Model	339
5.32	State Variables for VISCOELASTIC SWANSON Model	344
5.33	Material models documented in Sierra/SolidMechanics 4.46 Capabilities in Development	345
5.34	State Variables for TRACTION DECAY Surface Model	346
5.35	State Variables for TVERGAARD HUTCHINSON Surface Model	348
5.36	State Variables for THOULESS PARMIGIANI Surface Model	350
6.1	Default moduli ratios for the various effective moduli models.	370
6.5	Variables For Cylindrical Joint	440
9.1	Global Energy Variables	766
9.2	Other Global Variables For All Analyses	767
9.3	Global Variables for Rigid Bodies	767
9.4	Nodal Variables for All Analyses	768
9.5	Nodal Variables for Implicit Analyses	768
9.6	Nodal Variables for Shells and Beams	769
9.7	Nodal Variables for Spot Welds	770
9.8	Nodal Variables for Contact	771
9.9	Element Variables for All Elements	774
9.10	Element Variables for Solid Elements	776
9.11	Element Variables for Membranes	777
9.12	Element Variables for Shells	778
9.13	Element Variables for Trusses	779
9.14	Element Variables for Cohesive Elements	779
9.15	Element Variables for Nonlinear Beams	780
9.16	Element Variables for Springs	780

9.17 Face Variables	780
11.1 Subroutine Input Parameters	800
11.2 Subroutine Output Parameters	801
11.3 aupst_get_real_param Arguments	803
11.4 aupst_get_integer_param Arguments	804
11.5 aupst_get_string_param Arguments	805
11.6 aupst_evaluate_function Arguments	806
11.7 aupst_get_time Argument	806
11.8 aupst_check_node_var Arguments	808
11.9 aupst_check_elem_var Arguments	809
11.10aupst_get_node_var Arguments	810
11.11aupst_get_elem_var Arguments	811
11.12aupst_get_elem_var_offset Arguments	812
11.13aupst_put_node_var Arguments	813
11.14aupst_put_elem_var Arguments	814
11.15aupst_put_elem_var_offset Arguments	815
11.16aupst_check_global_var Arguments	817
11.17aupst_get_global_var Arguments	817
11.18aupst_put_global_var Arguments	818
11.19Topologies Used by Sierra/SM	819
11.20aupst_get_elem_topology Arguments	820
11.21aupst_get_elem_nodes Arguments	821
11.22aupst_get_face_topology Arguments	822
11.23aupst_get_face_nodes Arguments	823
11.24aupst_get_elem_centroid Arguments	824
11.25aupst_get_point Arguments	825
11.26aupst_get_proc_num Arguments	825
11.27Required aupst_preload_solver Parameters	850
11.28aupst_preload_solver Subroutine State Variables	853
11.29Required aupst_torque_calc Parameters	854
11.30Required aupst_trans_rot_velocity Parameters	855

C.1 Consistent Unit Sets	890
D.1 Constraint Enforcement Order	891
G.1 Required Variables For Consistent Model Initialization	902

Chapter 1

Introduction

Sierra/SolidMechanics (Sierra/SM) is a three-dimensional solid mechanics code with a versatile element library, nonlinear material models, large deformation capabilities, and contact. It is built on the SIERRA Framework [1, 2, 3, 4]. SIERRA provides a data management framework in a parallel computing environment that allows the addition of capabilities in a modular fashion. Contact capabilities are parallel and scalable.

This document provides information about the functionality in Sierra/SM and the command structure required to access this functionality in a user input file. This document is divided into chapters based primarily on functionality. For example, the command structure related to the use of various element types is grouped in one chapter; descriptions of material models are grouped in another chapter.

Sierra/SM provides both explicit transient dynamics and implicit quasistatics and dynamics capabilities. Both the explicit and implicit modules are highly scalable in a parallel computing environment. In the past, the explicit and implicit capabilities were provided by two separate codes, known as Presto and Adagio, respectively. These capabilities have been consolidated into a single code. The executable is named Adagio, but it provides the full suite of solid mechanics capabilities, for both implicit and explicit. The Presto executable has been disabled as a consequence of this consolidation.

Important references for both the implicit and explicit capabilities are given in the references section at the end of this chapter. Adagio was preceded by the codes JAC and JAS3D; JAC is described in [5]; JAS3D is described in [6]. Presto was preceded by the code Pronto3D. Pronto3D is described in [7] and [8]. Some of the fundamental nonlinear technologies used by Sierra/SM are described in [9, 10, 11]. Sierra/SM uses the Exodus II database and the XDMF database; Exodus II is more commonly used than XDMF. (Other options may be added in the future.) The Exodus II database format is described in [12], and the XDMF database format is described in [13]. The reference document for ACME [14] provides important information about contact. ACME is a third-party library for contact search and enforcement.

One of the key concepts for the command structure in the input file is a concept called *scope*. A detailed explanation of scope is provided in Section 1.3. Most of the command lines in Chapter 2 are related to a certain scope rather than to some particular functionality.

1.1 Document Overview

This document describes how to create an input file for Sierra/SM. Highlights of the document contents are as follows:

- Chapter 1 presents the overall structure of the input file, including conventions for the command descriptions, style guidelines for file preparation, and naming conventions for input files that reference the Exodus II database [12]. The chapter also gives an example of the general structure of an input file that employs the concept of scope.
- Chapter 2 explains some of the commands that are general to various applications based on the SIERRA Framework. These commands define scopes, functions, and coordinate systems, and they let you set up some of the main time control parameters (begin time, end time, time blocks) for your analysis. (Time control and time step control are discussed in more detail in Chapters 3 and 4.) Other capabilities documented in this chapter are available for calculating element distortion and for activating and deactivating functionality at different times throughout an analysis.
- Chapter 3 describes how to set the start time, end time, and time blocks for an explicit dynamic analysis. This chapter also discusses various options for controlling the critical time step for transient dynamics.
- Chapter 4 discusses the multilevel, nonlinear iterative solver used for implicit calculations. This chapter also describes how to set start time, end time, and time blocks for an implicit analysis.
- Chapter 5 describes material models that can be used in conjunction with the elements. Most of the material models can be used for both explicit and implicit analyses. Even though a material model can be used by both cases, it may be that the use of the material model is better suited for one type of analysis. For example, a material model set up to characterize behavior over a long time would be better suited for use in implicit analyses than in explicit analyses. In such cases, this will be noted. Chapter 5 also discusses the application of temperature to a mesh and the computation of thermal strains (isotropic and anisotropic).
- Chapter 6 lists the available elements and describes the commands used to access the various options for the elements. Most elements can be used for both explicit and implicit analyses. If that is not the case, a note is made to that effect.
Chapter 6 also includes descriptions of the commands for mass property calculations, element death, and mesh rebalancing. “Element-like” capabilities and rigid bodies are also discussed in Chapter 6.
- Chapter 7 documents how to use kinematic boundary conditions, force boundary conditions, initial conditions, and specialized boundary conditions.
- Chapter 8 discusses how to define interactions of contact surfaces.
- Chapter 9 details the various options for obtaining output.
- Chapter 10 documents special modeling techniques.
- Chapter 11 provides an overview of the user subroutine functionality.
- Chapter 12 documents how to perform transfers between procedures.
- Appendix A provides a sample input file for an explicit dynamic analysis of 16 lead spheres being crushed together inside a steel box. Appendix B provides a sample input file for an implicit, quasistatic analysis of an eraser being pulled across a surface. These problems both

emphasize large deformation and contact.

- The index allows you to find information about command blocks and command lines. In general, single-level entries identify the page where the command syntax appears, with discussion following soon thereafter—on the same page or on a subsequent page. Page ranges are not provided in this index. Some entries consist of two or more levels. Such entries are typically based on context, including such information as the command blocks in which a command line appears, the location of the discussion related to a particular command line, and tips on usage. The electronic version of this document contains hyperlinked entries from the page numbers listed in the index to the text in the body of the document.

1.2 Running The Code

The command to run a basic serial Sierra/SM analysis is:

```
sierra adagio -i sierra_input.i
```

The command to run a basic parallel Sierra/SM analysis is:

```
sierra -j 4 adagio -i sierra_input.i
```

The sierra command also optionally takes many more options to specify number of processors, queues to use, output log file names, etc. See the sierra command documentation for a full description of capabilities. The command to generate the sierra command documentation is:

```
sierra --help
```

The command to generate adagio specific options is:

```
sierra adagio --help
```

Note sierra script command line options have to be placed before the adagio command.

1.3 Overall Input Structure

Sierra/SM is one of many mechanics codes built on the SIERRA Framework. The SIERRA Framework provides the capability to perform multiphysics analyses by coupling together SIERRA codes appropriate for the mechanics of interest. Input files may be set up for analyses using only Sierra/SM, or they may be set up to couple Sierra/SM and one or more other SIERRA analysis codes. For example, you might run an implicit Sierra/SM analysis to compute a stress state, and then use the results of this analysis as initial conditions for an explicit analysis. For a Sierra multiphysics analysis, the commands for all physics modules used in the analysis will all be in the same input file. Therefore, the input for Sierra/SM reflects the fact that it could be part of a multiphysics analysis. Sierra/SM coupled with the Sierra package Aria is Arpeggio. Sierra/SM coupled with the Sierra/SD is Tempo. And Fortissimo is Sierra/SM coupled with the Sandia package CTH.

Sierra input file syntax uses ideas from programming languages. A multiphysics analysis is defined using *scope*, i.e., the extent of the subject matter that part of the input file deals with, to group similar commands. Examples of the Sierra input files are given in this section, section 1.7, and appendices A and B.

The broadest scope in the input file is the SIERRA scope. The SIERRA scope contains information that can be shared among different physics. Examples of physics information that can be shared are definitions of functions and materials. For example, in a coupled implicit/explicit Sierra/SM analysis, both the implicit and explicit components could reference functions to define time histories for boundary conditions or stress-strain curves. Material model definitions could also be shared in a similar manner.

A scope may contain another scope. Within a SIERRA scope the overall analysis from the start time to the end time is controlled by the `procedure` scope. In a multiphysics analysis, a SIERRA scope may contain multiple procedures.

The `procedure` scope may contain a `region` scope. The region scope controls a single time step. Inside the procedure scope (but outside of the region scope) are commands that set the start time and the end time for the analysis.

Inside the region scope for Sierra/SM are such things as definitions for boundary conditions and contact. In a multiphysics analysis, there would be more than one region. In a coupled implicit/explicit solid mechanics example, there would be both a Presto region and an Adagio region, each within its respective procedures. The definitions for boundary conditions and contact and the mesh specification for explicit dynamics would appear in the Presto region; the definitions for boundary conditions and contact and the mesh specification for the implicit analysis would appear in the Adagio region.

Scope is defined by a command block. The command blocks group together command lines or other command blocks that share a similar functionality. A command block will begin with an input line that has the word `BEGIN` and end with an input line that has the word `END`. The SIERRA scope, for example, is defined by a command block that begins with an input line of the following form:

```
BEGIN SIERRA my_problem
```

The two character strings `BEGIN` and `SIERRA` are the key words for this command block. An input

line defining a command block or a command line will have one or more key words. The string `my_problem` is a user-specified name for this SIERRA scope. The SIERRA scope is terminated by an input line of the following form:

```
END SIERRA my_problem
```

In the above input line, `END` and `SIERRA` are the key words to end this command block. SIERRA scope termination may be shortened to

```
END
```

as discussed in section 1.5.9. Similar input lines are used to define the procedure and region scopes. Boundary conditions are another example where a scope is defined. A particular instance of a boundary condition for a prescribed displacement boundary condition is defined with a command block. The command block for the boundary condition begins with an input line of the form:

```
BEGIN PRESCRIBED DISPLACEMENT
```

and ends with an input line of either of these forms:

```
END PRESCRIBED DISPLACEMENT
```

```
END
```

Command lines appear within the command blocks. The command lines typically have the form `keyword = value`, where `value` can be a real, an integer, or a string. In the previous example of the prescribed displacement boundary condition, there would be command lines inside the command block that are used to set various values. For example, the boundary condition might apply to all nodes in node set 10, in which case there would be a command line of the following form:

```
NODE SET = nodelist_10
```

If the prescribed displacement were to be applied along a given component direction, there would be a command line of this form:

```
COMPONENT = X
```

The form above would specify that the prescribed displacement would be in the x -direction. Finally, if the displacement magnitude is described by a time history function with the name `my_function`, there would be a command line of this form:

```
FUNCTION = my_function
```

Functions are discussed in sections 2.1.5 and 2.1.6.

The command block for the boundary condition with the appropriate command lines would appear as follows:

```
BEGIN PRESCRIBED DISPLACEMENT
  NODE SET = nodelist_10
  COMPONENT = X
  FUNCTION = my_function
END PRESCRIBED DISPLACEMENT
```

It is possible to have a command line with the same key words appearing in different scopes. For

example, we might have a command line identified by the word `TYPE` (introduced in section 2.1.5) in two or more different scopes. The command line would perform different functions based on the scope in which it appeared, and the associated value could be different in the two locations.

The input lines are read by a parser that searches for recognizable key words. If the key words in an input line are not in the list of key words used by Sierra/SM to describe command blocks and command lines, the parser will generate an error. A set of key words defining a command line or command block for Sierra/SM that is not in the correct scope will also cause a parser error. For example, the key words `STEP INTERVAL` define a valid command line in the scope of the `TIME CONTROL` command block. However, if this command line was to appear in the scope of one of the boundary conditions, it would not be in the proper scope, and the parser would generate an error. Once the parser has an input line with any recognizable key words in the proper scope, a method can be called that will handle the input line.

There is an initial parsing phase that checks only the parser syntax. If the parser encounters a command line it cannot parse within a certain scope, the parser will indicate it cannot recognize the command line and will list the various command lines that can appear within that scope. The initial parsing phase will catch errors such as the one described in the previous paragraph (a command line in the wrong scope). It will also catch misspelled key words.

The initial parsing catches some but not all types of errors. Some errors not detected in the initial parsing phase are caught afterwards and are handled one at a time. One such example is that initial parsing will not catch a command line value that is out of command line's specified range. A second example is a combination of command lines within a command block that is not allowed.

1.4 Conventions for Command Descriptions

The parts of an input file built into Sierra can be hard to distinguish from the user supplied parts. For example in section 1.3, the user supplied only the name `my_problem` for the `SIERRA` scope and the name `my_function` for a function. This section presents the syntax used in this document to describe input files.

A note about continuation characters. Many individual command lines discussed in the text appear on several text lines. In the text of this document, the continuation symbols that are used to continue lines in an actual input file (`\#` and `\$`, Section 1.5.2) are not used for those instances where the description of the command line appears on several text lines. The description of command lines will indicate all the key words, delimiters, and values that constitute a complete command line. As an example, the `DEFINE POINT` command line (Section 2.1.7) is presented in the text as follows:

```
DEFINE POINT <string>point_name WITH COORDINATES
      <real>value_1 <real>value_2 <real>value_3
```

If the `DEFINE POINT` command line were used as a command line in an input file and spread over two input lines, it would appear, with actual values, as follows:

```
DEFINE POINT center WITH COORDINATES \#
      10.0 144.0 296.0
```

In the above example, the `\#` symbol implies the first line is continued onto the second line.

1.4.1 Key Words

The key word or key words for a command are shown in uppercase letters. For actual input, you can use all uppercase letters for the key words, all lowercase letters for the key words, or some combination of uppercase and lowercase letters for the key words.

1.4.2 User-Specified Input

The input that you supply is typically shown in lowercase letters. (Occasionally, uppercase letters may be used for user input for purposes of clarity or in examples.) The user-supplied input may be a real number, an integer, a string, or a string list. For the command descriptions, a type appears before the user input. The type (real, integer, string, string list) description is enclosed by angle brackets, `<>`, and precedes the user-supplied input. For example:

```
<real>value
```

indicates that the quantity `value` is a real number. For the description of an input command, you would see:

```
FUNCTION = <string>function_name
```

Your input would be

```
FUNCTION = my_name
```

if you have specified a function name called `my_name`.

Valid user input consists of:

<code><integer></code>	Integer data is a single integer number.
<code><real></code>	Real data is a single real number. It may be formatted with the usual conventions, such as 1234.56 or 1.23456e+03.
<code><string></code>	String data is a single string.
<code><string list></code>	A string list consists of multiple strings separated by white space, a comma, a tab, or white space combined with a comma or a tab.

1.4.3 Optional Input

Anything in an input line that is enclosed by square brackets, [], represents optional input within the line. This convention is not used to identify optional input lines. Any command line that is optional (in its entirety) will be described within the text.

1.4.4 Default Values

A value enclosed by parentheses, (), appearing after the user input denotes the default value. For example:

```
SCALE FACTOR = <real>scale_factor(1.0)
```

implies the default value for `scale_factor` is 1.0. Any value you specify will overwrite the default.

For your actual input file, you may simply omit a command line if you want to use the default value associated with the command line. For example, there is a `TIME STEP SCALE FACTOR` command line used to set one of the time control parameters; the parameter for this command line has a default value of 0.9. If you want to use the default value of 0.9 for this parameter, you do not have to include the `TIME STEP SCALE FACTOR` command line in the `TIME CONTROL` command block.

1.4.5 Multiple Options for Values

Quantities separated by the | symbol indicate that one and only one of the possible choices must be selected. For example:

```
EXPANSION RADIUS = <string>SPHERICAL|CYLINDRICAL
```

implies that expansion radius must be defined as `SPHERICAL` or `CYLINDRICAL`. One of the values must appear. This convention also applies to some of the command options within a begin/end block. For example:

```
SURFACE = <string>surface_name|
NODE SET = <string>nodelist_name
```

in a command block specifies that either a surface or a node set must be specified.

Quantities separated by the / symbol can appear in any combination, but any one quantity in the sequence can appear only once. For example,

```
COMPONENTS = <string>X/Y/Z
```

implies that components can equal any combination of X, Y, and Z. Any value (X or Y or Z) can appear at most once, and at least one value of X, Y, or Z must appear. Some examples of valid expressions in this case are as follows:

```
COMPONENTS = Z
```

```
COMPONENTS = Z X
```

```
COMPONENTS = Y X Z
```

```
COMPONENTS = Z Y X
```

An example of an invalid expression would be the following:

```
COMPONENTS = Y Y Z
```

1.4.6 Known Issues and Warnings

Where there are known issues with the code, these are documented in the following manner:



Known Issue: A description of the known issue with the code would be provided here.

Similarly, warnings regarding usage of code features that are not defective, but must be used with care because of their nature, are documented as follows:



Warning: A description of the warning related to the usage of a code feature would be provided here.

1.4.7 Commands Applicable to Only Explicit or Implicit Analyses

As mentioned previously, Sierra/SM has the ability to run either explicit dynamic analyses and implicit quasistatic and dynamic analyses. This User's Guide documents all features of the code for either of these analysis types. In most cases, the features available for explicit analyses are also available for implicit analysis, and vice versa. However, there are some capabilities that are only available for one type of analysis. These are denoted in this document with the following icons in the margin.



Features and commands only available for explicit dynamics have this icon in the margin next to their documentation.



Features and commands only available for implicit dynamics and quasistatics have this icon in the margin next to their documentation.

In listings of command lines and blocks, smaller versions of these icons are used to denote com-

mands only available for explicit or implicit analyses. These appear in the margin next to either an individual command line, or next to the opening line of a command block, as shown in these examples:

 **Explicit**

```
#Example explicit command line:  
DEATH ON INVERSION = OFF|ON(OFF)
```

 **Implicit**

```
#Example implicit command line:  
CAPTURE TOLERANCE = <real>cap_tol
```

 **Explicit**

```
#Example explicit command block:  
BEGIN LINE WELD  
...  
END [LINE WELD]
```

 **Implicit**

```
#Example implicit command block:  
BEGIN SOLVER  
...  
END [SOLVER]
```

1.5 Style Guidelines

This section gives information that will affect the overall organization and appearance of an input file. It also contains recommendations that will help construct input files that are readable and easy to proof.

1.5.1 Comments

A comment is anything between the # symbol or the \$ symbol and the end-of-line. If the first non-blank character in a line is a # or \$, the entire line is a comment line. You can also place a # or \$ (preceded by a blank space) after the last character in an input line used to define a command block or command line.

1.5.2 Continuation Lines

An input line can be continued by placing a \# pair of characters (or \\$) at the end of the line. The following line is then taken to be a continuation of the preceding line that was terminated by the \# or \\$. Everything after the line-continuation pair of characters is discarded, including the end-of-line.

1.5.3 Capital and Lower Case Letters

Almost all the character strings in the input lines are case insensitive. For example, the `BEGIN SIERRA` key words could appear as:

```
BEGIN SIERRA
begin sierra
Begin Sierra
```

You could specify a `SIERRA` command block with:

```
BEGIN SIERRA BEAM
```

and terminate the command block with this input line:

```
END SIERRA beam
```

Case is important only for file name specifications. If you have defined a restart file with uppercase and lowercase letters and want to use this file for a restart, the file name you use to request this restart file must exactly match the original definition you chose.

1.5.4 Commas and Tabs

Commas and tabs in input lines are ignored.

1.5.5 Blank Spaces

We recommend that everything be separated by blank spaces. For example, a command line of the form

```
node set = nodelist_10
```

is recommended over the these forms:

```
node set= nodelist_10
node set =nodelist_10
```

Both of the above two lines are correct, but it is easier to check the first form (the equal sign surrounded by blank space) in a large input file.

The parser will accept the following line:

```
BEGIN SIERRABEAM
```

However, it is harder to check this line for the correct spelling of the key words and the intended SIERRA scope name than this line:

```
BEGIN SIERRA BEAM
```

It is possible to introduce hard-to-detect errors because of the way in which the blank spaces are handled by the command parser. Suppose you type

```
begin functions my_func
```

rather than the following correct form:

```
begin function my_func
```

For the incorrect form of this command line (in which `functions` is used rather than `function`), the parser will generate a string name of

```
s my_func
```

for the function name rather than the following expected name:

```
my_func
```

If you attempt to use a function named `my_func`, the parser will generate an error because the list of function names will include `s my_func` but not `my_func`.

1.5.6 General Format of the Command Lines

In general, command lines have the following form:

```
keyword = value
```

This pattern is not always followed, but it describes the vast majority of the command lines.

1.5.7 Delimiters

The delimiter used throughout this document is “=” (the equal sign). Typically, but not always, the = separates key words from input values in a command line. Consider this command line:

```
COMPONENTS = X
```

Here, the key word `COMPONENTS` is separated from its value, a string in this case, by the `=`. Some command lines do allow for other delimiters. The use of these alternate delimiters is not consistent, however, throughout the various command lines. (This lack of consistency has the potential for introducing errors in this document as well as in your input.) The `=` provides a strong visual cue

for separating key words from values. By using the = as a delimiter, it is easier to proof your input file. It also makes it easier to do “cut and paste” operations. If you accidentally delete =, it is easier to detect than accidentally removing part of one of the other delimiters that could be used.

1.5.8 Order of Commands

Commands have no ordering requirements. Both the input sequence:

```
BEGIN PRESCRIBED DISPLACEMENT
  NODE SET = nodelist_10
  COMPONENT = X
  FUNCTION = my_function
END PRESCRIBED DISPLACEMENT
```

and the input sequence:

```
BEGIN PRESCRIBED DISPLACEMENT
  FUNCTION = my_function
  COMPONENT = X
  NODE SET = nodelist_10
END PRESCRIBED DISPLACEMENT
```

are valid, and they produce the same result. Remember, that command lines and command blocks must appear in the proper scope.

1.5.9 Abbreviated END Specifications

It is possible to terminate a command block without including the key word or key words that identify the block. You could define a specific instance of the prescribed displacement boundary condition with:

```
BEGIN PRESCRIBED DISPLACEMENT
```

and terminate it simply with:

```
END
```

as opposed to this specification:

```
END PRESCRIBED DISPLACEMENT
```

Both the short termination (END only) and the long termination (END followed by identification, or name, of the command block) are valid. It is recommended that the long termination be used for any command block that becomes large. The RESULTS OUTPUT command block described in later chapters can become fairly lengthy, so this is probably a good place to use the long termination. For most boundary conditions, the command block will typically consist of five lines. In such cases, the short termination can be used. Using the long termination for the larger command blocks will make it easier to proof your input files. If you use the long termination, the text following the END key word must exactly match the text following the BEGIN key word. You could not have BEGIN PRESCRIBED DISPLACEMENT paired with an END PRESCRIBED DISPL to define the beginning and ending of a command block.

1.5.10 Indentation

When constructing an input file, it is useful, but not required, to indent a scope that is nested inside another scope. Command lines within a command block should also be indented in relation to the lines defining the command block. This will make it easier to construct the input file with everything in the correct scope and with all the command blocks in the correct structure.

1.5.11 Aprepro Text Processing

Aprepro is used for input deck text preprocessing. Generally any text inside curly brackets { } will be interpreted by aprepro. This include text inside of comments. See [15] for more information. The sierra script can run aprepro automatically on an input file by adding a command line option as shown in this example:

```
sierra -a adagio -i sierra_input.i
```

Note, if there is text in a file surrounded by brackets { }, and you do not want it to be processed by aprepro, it can be “escaped,” or turned off for that text block, with the ‘\textbackslash ’ character. For example

```
# Include an external file with aprepro
#   {include('`matProps.inp'`)}

# Turn aprepro off processing so that matProps is not included
#   \{include('`matProps.inp'`)}\}
```

1.5.12 Including Files

External text files containing input commands can be included at any point in the Sierra/SM input file using the aprepro `include` command. This command can be used in any context in the input file. To use this command, simply use the command `include` command followed by the name of the file to be included. For example, the command:

```
{include("displacement_history.i")}
```

would include the `displacement_history.i` as if the contents of that file were placed in the position that it is included in the input file. The included file is contained in the standard echo of the input provided at the beginning of the log file. If requested, aprepro substitution will occur inside of the included text. Note, the user must use double quotes around the name of the text file. If single quotes are used, aprepro will not be able to find the file.

1.6 Exodus II Database Naming Conventions

When the mesh file has an Exodus II format, there are three basic conventions that apply to user input for various command lines. The conventions concern side sets (surfaces), node sets and blocks.

First, for a mesh file with the Exodus II format, the Exodus II side set is referenced as a surface. In SIERRA, a surface consists of element faces plus all the nodes and edges associated with these faces. A surface definition can be used not only to select a group of faces but also to select a group of edges or a group of nodes that are associated with those faces. In the case of boundary conditions, a surface definition can be used not only to apply boundary conditions that typically use surface specifications (pressure) but also to apply boundary conditions for what are called nodal boundary conditions (fixed displacement components). For nodal boundary conditions that use the surface specification, all the nodes associated with the faces on a specific surface will have this boundary condition applied to them. The specification for a surface identifier in the following chapters is `surface_name`. It typically has the form `surface_integerid`, where `integerid` is the integer identifier for the surface. If the side set identifier is 125, the value of `surface_name` would be `surface_125`. It is also possible to generate an alias for the side set¹ and use this for `surface_name`. If `surface_125` is aliased to `outer_skin`, then `surface_name` becomes `outer_skin` in the actual input line. It is also possible to name a surface in some mesh generation programs and that name can be used in the input file.

Second, for a mesh file with the Exodus II format, the Exodus II node set is still referenced as a node set. A node set can be used only for cases where a group of nodes needs to be defined. The specification for a node set identifier in the following chapters is `nodelist_name`. It typically has the form `nodelist_integerid`, where `integerid` is the integer identifier for the node set. If the node set number is 225, the value of `nodelist_name` would be `nodelist_225`. It is also possible to generate an alias for the node set and use this for `nodelist_name`. If `nodelist_225` is aliased to `inner_skin`, then `nodelist_name` becomes `inner_skin` in the actual input line. It is also possible to name a nodelist in some mesh generation programs and that name can be used in the input file.

Third, an element block is referenced as a block. The specification for an element block identifier in the following chapters is `block_name`. It typically has the form `block_integerid`, where `integerid` is the integer identifier for the block. If the element block number is 300, the value of `block_name` would be `block_300`. It is also possible to generate an alias for the block and use this for `block_name`. If `block_300` is aliased to `big_chunk`, then `block_name` becomes `big_chunk` in the actual input line. It is also possible to name an element block in some mesh generation programs and that name can be used in the input file.

A group of elements can also be used to select other mesh entities. In SIERRA, a block consists of elements plus all the faces, edges, and nodes associated with the elements. The block and surface concepts are similar in that both have associated derived quantities. Chapters 7 and 8 show how this concept of derived quantities is used in the input command structure.

¹See the `ALIAS` command in Section 6.1.2

1.7 Major Scope Definitions for an Input File

The typical input file will have the structure shown in the two examples below. The major scopes—SIERRA, procedure, and region—are delineated with input lines for command blocks. Comment lines are included that indicate some of the key scopes that will appear within the major scopes. Note the indentation used to clarify scopes. An example of the structure an explicit dynamic (Presto) input file is shown:

Explicit

```
BEGIN SIERRA <string>some_name
#
# All command blocks and command lines in the SIERRA
# scope appear here. The PROCEDURE PRESTO command
# block is the beginning of the next scope.
#
# function definitions
# material descriptions
# description of mesh file
#
BEGIN PRESTO PROCEDURE <string>procedure_name
#
# time step control
#
BEGIN PRESTO REGION <string>region_name
#
# All command blocks and command lines in the
# region scope appear here
#
# specification for output of result
# specification for restart
# boundary conditions
# definition of contact
#
END [PRESTO REGION <string>region_name]
END [PRESTO PROCEDURE <string>procedure_name]
END [SIERRA <string>some_name]
```

This is an example of an implicit (Adagio) input file:

Implicit

```
BEGIN SIERRA <string>some_name
#
# All command blocks and command lines in the SIERRA
# scope appear here. The PROCEDURE ADAGIO command
# block is the beginning of the next scope.
#
# function definitions
# material descriptions
# description of mesh file
```

```

#
BEGIN ADAGIO PROCEDURE <string>procedure_name
#
# time step control
#
BEGIN ADAGIO REGION <string>region_name
#
# All command blocks and command lines in the
# region scope appear here
#
# solver commands
# specification for output of result
# specification for restart
# boundary conditions
# definition of contact
#
END [ADAGIO REGION <string>region_name]
END [ADAGIO PROCEDURE <string>procedure_name]
END [SIERRA <string>some_name]

```


1.8 Input/Output Files

The primary user input to Sierra/SM is the input file introduced in this chapter. Throughout this document, we explain how to construct a valid input file. Be aware that Sierra/SM also processes many other types of input files and produces a variety of output files. These additional files are also discussed in this document where applicable. Figure 1.1 shows a simple schematic diagram of the various input and output files.

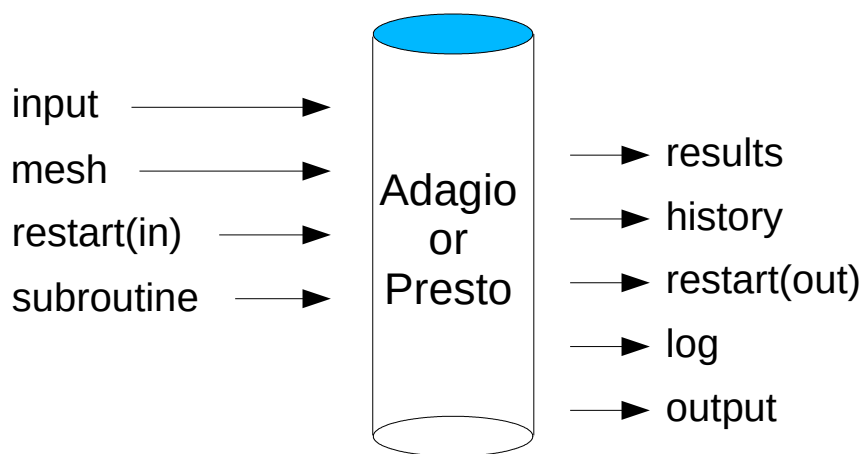


Figure 1.1: Input/output files

As shown in Figure 1.1, Sierra/SM uses the input file, mesh files, restart files, and user subroutine files. The input file, which is required, is a set of valid Sierra/SM command lines. Another required input is a mesh file, which provides a description of the finite element mesh for the object being analyzed. Restart and user subroutine files are optional inputs. The restart functionality lets you break an analysis from the start time to the termination time into a sequence of runs. The files generated by the restart functionality contain a complete state description for a problem at various analysis times, which we will refer to as restart times. You can restart Sierra/SM at any of these restart times because the complete state description is known (see Chapter 9). The user subroutine files let you build and incorporate specialized functionality into Sierra/SM (Chapter 11).

As also shown in Figure 1.1, Sierra/SM can generate a number of files. These include results files, history files, restart files, a log file, and an output file. Typically, only the log file and the output file are produced automatically. Generation of the other types of files is based on user settings in the input file for the particular output desired. Results files provide the values of global variables, element variables, and node variables at specified times (see Chapter 9). History files will also provide values of global variables, element variables, and node variables at specified times (see Chapter 9). History files are set up to provide a specific value at a specific node, for example, whereas results files provide a nodal value for large subsets of nodes or, more typically, all nodes. History files provide a more limited set of information than results files. As noted above, restart files can be generated at various analysis times. The log file contains a variety of information such as the Sierra/SM version number, a listing of the input file, initialization information, some model information (mass, critical time steps for element blocks, etc.), and information at various time steps. At every n^{th} step, where n is user selected, the log file gives the current analysis time; the

current time step; the kinetic, internal, and external energies; the error in the energy; and computing time information. You can monitor step information in the log file to gain information about how your analysis is progressing. The output file contains error information.

1.9 Obtaining Support

Support for all SIERRA Mechanics codes, including Sierra/SM, can be obtained by contacting the SIERRA Mechanics user support team by email at sierra-help@sandia.gov. Users on the Sandia Restricted Network may consult the Computational Simulation web pages (<http://compsim.sandia.gov/compsim/>) for "Getting Started" information and basic troubleshooting guidance of general Sierra issues.

1.9.1 Helpful Information to Include when Requesting Support

When making an inquiry to sierra-help, providing the following information will be helpful to get the most useful and timely response.

- A description of the problem, such as error message obtained or what behavior was observed versus what was expected.
- The input deck, log file, and mesh files associated with the run. If these files are under 4MB they can be sent directly via email to sierra-help. Alternatively the path to the directory with the run in question may be given. If the files related to the run are large or have access restrictions contact sierra-help to determine the best method to provide the user support team the relevant information about the run.
- Any information that may help the development team narrow down the problem. Such as which input deck command was added to an analysis that caused an error, if the problem ran in a previous version of the code, or if the same error was seen on other analyses.
- The priority of the request as it relates to your analysis. For example, is the problem a show-stopper for the analysis or is it merely an inconvenience? Is there a deadline looming that makes resolution of your issue a high priority?

References

- [1] H.C. Edwards and J.R. Stewart. Sierra framework for developing advanced parallel mechanics applications. In L.T. Biegler, editor, *Large-Scale PDE-Constrained Optimization*, 2003.
- [2] H.C. Edwards and J.R. Stewart. A framework approach for developing parallel adaptive multiphysics applications. *Finite Elements in Analysis and Design*, 40(12), 2004.
- [3] H.C. Edwards and J.R. Stewart. SIERRA: A software environment for developing complex multi-physics applications. In K.J. Bathe, editor, *First MIT Conference on Computational Fluid and Solid Mechanics*, pages 1147–1150, Amsterdam, 2001. Elsevier.
- [4] J.R. Koteris, A.S. Gullerud, V.L. Porter, W.M. Scherzinger, and K.H. Brown. PRESTO: Impact dynamics with scalable contact using the SIERRA framework. In K.J. Bathe, editor, *First MIT Conference on Computational Fluid and Solid Mechanics*, pages 294–296, Amsterdam, 2001. Elsevier.
- [5] J.H. Biffle. JAC – a two-dimensional finite element computer program for the non-linear quasi-static response of solids with the conjugate gradient method. Technical Report SAND81-0998, Sandia National Laboratories, Albuquerque, NM, 1984. [pdf](#).

- [6] M.L. Blanford, M.W. Heinstein, and S.W. Key. JAS3D – a multi-strategy iterative code for solid mechanics analysis users’ instructions, release 2.0, September 2001.
- [7] L.M. Taylor and D.P. Flanagan. Pronto3D: A three-dimensional transient solid dynamics program. Technical Report SAND87-1912, Sandia National Laboratories, Albuquerque, NM, March 1989. [pdf](#).
- [8] S.W. Attaway, K.H. Brown, F.J. Mello, M.W. Heinstein, J.W. Swegle, J.A. Ratner, and R.I. Zadoks. Pronto3D user’s instructions: A transient dynamic code for nonlinear structural. Technical Report SAND98-1361, Sandia National Laboratories, Albuquerque, NM, 1998. [pdf](#).
- [9] T.A. Laursen, S.W. Attaway, and R.I. Zadoks. SEACAS theory manuals: Part I. problem formulation in nonlinear solid mechanics. Technical Report SAND98-1760/1, Sandia National Laboratories, Albuquerque, NM, 1998. [pdf](#).
- [10] T.A. Laursen, S.W. Attaway, and R.I. Zadoks. SEACAS theory manuals: Part II. nonlinear continuum mechanics. Technical Report SAND98-1760/2, Sandia National Laboratories, Albuquerque, NM, 1998. [pdf](#).
- [11] T.A. Laursen, S.W. Attaway, and R.I. Zadoks. SEACAS theory manuals: Part III. finite element analysis in nonlinear solid mechanics. Technical Report SAND98-1760/3, Sandia National Laboratories, Albuquerque, NM, 1999. [pdf](#).
- [12] L.A. Schoof and V.R. Yarberr. EXODUS II: A finite element data model. Technical Report SAND92-2137, Sandia National Laboratories, Albuquerque, NM, September 1994. [pdf](#). See also documentation available at EXODUS II Sourceforge page. [link](#).
- [13] The eXtensible Data Model and Format (xdmf). [link](#).
- [14] K.H. Brown, R.M. Summers, M.W. Glass, A.S. Gullerud, M.W. Heinstein, and R.E. Jones. ACME: Algorithms for contact in a multiphysics environment, API version, 1.0. Technical Report SAND2001-3318, Sandia National Laboratories, Albuquerque, NM, 2001. [pdf](#).
- [15] G.D. Sjaardema. APREPRO: An algebraic preprocessor for parameterizing finite element analyses. Technical Report SAND92-2291, Sandia National Laboratories, Albuquerque, NM, 1999. [pdf](#).

Chapter 2

General Commands

The commands described in this section appear in the SIERRA or procedure scope or control general functionality in Sierra/SM.

2.1 SIERRA Scope

These commands are used to set up some of the fundamentals of the Sierra/SM input. The commands are physics independent, or at least can be shared between physics. The commands lie in the SIERRA scope, not in the procedure or region scope.

2.1.1 SIERRA Command Block

```
BEGIN SIERRA <string>name
#
# All other command blocks and command lines
# appear within the SIERRA scope defined by
# begin/end sierra.
#
END [SIERRA <string>name]
```

All input commands must occur within a SIERRA command block. The syntax for beginning the command block is:

```
BEGIN SIERRA <string>name
```

and for terminating the command block is as follows:

```
END [SIERRA <string>name]
```

In these input lines, `name` is a name for the SIERRA command block. All other commands for the analysis must be within this command block structure. The name for the SIERRA command block is often a descriptive name that identifies the analysis. The name is not currently used anywhere else in the file and is completely arbitrary.

2.1.2 Title

```
TITLE <string list>title
```

To permit a fuller description of the analysis, the input has a `TITLE` command line for the analysis, where `title` is a text description of the analysis. The title is transferred to the results file.

2.1.3 Restart Control

Restart allows a user to resume an analysis by initializing all of its fields from a specific time of a previous simulation. For more background information on restart, see Section 9.6.

The time at which to restart an analysis can be specified in either the Sierra scope or the `RESTART DATA` command block in the region scope. If the restart time is specified in the Sierra scope, a `RESTART DATA` command block is still required to specify input and output restart data filenames. When performing a multi-physics simulation involving coupling Sierra/SM to another Sierra code, it is beneficial to specify the restart time in the Sierra scope. However, if the analysis involves only Sierra/SM, it is sufficient to specify a restart time only in the `RESTART DATA` command block.

If using restart commands in the Sierra scope, one of two restart command lines (`RESTART` or `RESTART TIME`) are used. The details of these two commands are described below:

2.1.3.1 Restart Time

```
RESTART TIME = <real>restart_time
```

The `RESTART TIME` command line is used to specify a time from which to restart the simulation. This restart option will pick the restart time on the restart file that is closest to the user-specified time in the `RESTART TIME` command line. If the user specifies a restart time greater than the last time written to a restart file, the last time written to the restart file is picked as the restart time. If this command is specified, a restart file must exist. If the input restart database does not exist, the code will exit with an error. For details of this feature and a use case example, see Section 9.6.1.3.

Using the `RESTART TIME` command requires the user to be more active in the management of the file names to prevent the overwriting of restart, results, and history files. To prevent the overwriting of existing restart files, an input restart file and an output restart file can be specified in the `RESTART DATA` command block. To prevent the overwriting of results and history files, the filenames must be changed in the respective output blocks. It is best to break the analysis into a sequence of 'restart runs' where each 'restart run' will consist of its own restart, results, and history files. For more information on overwriting of output files, see Chapter 9. If the user would prefer not to manage the overwriting of files, the automatic restart feature should be used. This feature is outlined below in Section 2.1.3.2.

2.1.3.2 Automatic Restart

```
RESTART = AUTOMATIC|AUTO
```

The `AUTOMATIC` or `AUTO` restart option automatically selects the last time written to the last restart file as the restart time. The automatic restart feature allows the user to restart runs with minimal changes to the input file. The only quantity that must be changed to move from one restart to another is the termination time. The `RESTART` command line manages the restart files to ensure any previous restart files are not overwritten. It also manages the results and history files to ensure

any previous results or history files are not overwritten as well. For details of this feature and a use case example, see Section 9.6.1.2.

If the `AUTOMATIC` option is used and the restart database does not exist, the analysis will start from the initial time and not read any restart data. Note, the path to the restart file must be carefully defined when using the `AUTOMATIC` option. If there is a typo in the path or file name that prevents the correct restart database from being found, the analysis will start from the beginning of the first time period without error.

See Section 9.6 for a full discussion of implementing the restart capability.

2.1.4 User Subroutine Identification

```
USER SUBROUTINE FILE = <string>file_name
```

This command line is a part of a set of commands that are used to implement the user subroutine functionality. The string `file_name` identifies the name of the file that contains the FORTRAN code of one or more user subroutines.

To understand how this command line is used, see Chapter 11.

2.1.5 XY Functions

```
BEGIN FUNCTION <string>function_name
  TYPE = <string>CONSTANT|PIECEWISE LINEAR|PIECEWISE CONSTANT|
    ANALYTIC|PIECEWISE ANALYTIC|MULTICOLUMN PIECEWISE LINEAR|
    PIECEWISE MULTIVARIATE
  EXPRESSION VARIABLE: <var_name> = NODAL|NODAL_VECTOR|
    NODAL_TENSOR|NODAL_SYM_TENSOR|ELEMENT|ELEMENT_VECTOR|
    ELEMENT_TENSOR|ELEMENT_SYM_TENSOR|FACE|
    GLOBAL <var_source>
  ABSCISSA = <string>abscissa_label
  ORDINATE = <string>ordinate_label
  X SCALE = <real>x_scale(1.0)
  X OFFSET = <real>x_offset(0.0)
  Y SCALE = <real>y_scale(1.0)
  Y OFFSET = <real>y_offset(0.0)
  ABSCISSA SCALE = <real>x_scale(1.0)
  ABSCISSA OFFSET = <real>x_offset(0.0)
  ORDINATE SCALE = <real>y_scale(1.0)
  ORDINATE OFFSET = <real>y_offset(0.0)
  BEGIN VALUES
    <real>x_1    <real>y_1
    <real>x_2    <real>y_2
    ...
    <real>x_n    <real>y_n
  END [VALUES]
  BEGIN EXPRESSIONS
    <real>x_1    <string>analytic_expression_1
```

```

    <real>x_2    <string>analytic_expression_2
    ...
    <real>x_n    <string>analytic_expression_n
END
DATA FILE = <string>fileName
    [X FROM COLUMN <int>xCol Y FROM COLUMN <int>yCol]

AT DISCONTINUITY EVALUATE TO <string>LEFT|RIGHT (LEFT)
EVALUATE EXPRESSION = <string>analytic_expression1;
    analytic_expression2; ...
DIFFERENTIATE EXPRESSION = <string>analytic_expression1;
    analytic_expression2; ...
EVALUATE FROM <real> TO <real> BY <real>
DEBUG = ON|OFF (OFF)
END [FUNCTION <string>function_name]

```

Many Sierra/SM features are driven by a user-defined description of the dependence of one variable on another. For instance, the prescribed displacement boundary condition requires the definition of a time-versus-displacement relation, and the thermal strain computations require the definition of a thermal-strain-versus-temperature relation. SIERRA provides a general method of defining these relations as functions using the `FUNCTION` command block, as shown above.

Many options exist for defining a general X versus Y function. More limited options exist for defining more complex functions of multiple variables. The more complex functions are described in Section [2.1.6](#).

There is no limit to the number of functions that can be defined. All function definitions must appear within the SIERRA scope.

A description of the various parts of the `FUNCTION` command block follows:

- The string `function_name` is a user-selected name for the function that is unique to the function definitions within the input file. This name is used to refer to this function in other locations in the input file.
- The `TYPE` command line has seven options to define the type of function. The value of this string can be `CONSTANT`, `PIECEWISE LINEAR`, `PIECEWISE CONSTANT`, `ANALYTIC`, `PIECEWISE ANALYTIC`, `MULTICOLUMN PIECEWISE LINEAR`, or `PIECEWISE MULTIVARIATE`.
- The `ABSCISSA` command line provides a descriptive label for the independent variable (x -axis) with the string `abscissa_label`. This command line is optional. (Note: Previous versions of SIERRA allowed the user to optionally add a scale factor and/or an offset via the `SCALE` and `OFFSET` keywords. These keywords have now been deprecated and their use will generate a parse error. Any desired scale factors and offsets need to be specified via the `X SCALE` and/or `X OFFSET` command lines, respectively.)
- The `ORDINATE` command line provides a descriptive label for the dependent variable (y -axis) with the string `ordinate_label`. This command line is optional. (Note: Previous versions of SIERRA allowed the user to optionally add a scale factor and/or an offset via the `SCALE` and `OFFSET` keywords. These keywords have now been deprecated and their use

will generate a parse error. Any desired scale factors and offsets need to be specified via the `Y SCALE` and/or `Y OFFSET` command lines, respectively.)

- The `X SCALE` command line sets the scale factor value for the abscissa, which has the following effect: $abscissa_{scaled,offset} = scale * (abscissa + offset)$. (Note: If no offset is specified via `X OFFSET`, $offset = 0.0$.) This command is not available for analytic or piecewise analytic functions. `ABSCISSA SCALE` is an alias for `X SCALE`.
- The `X OFFSET` command line sets the offset value for the abscissa, which has the following effect: $abscissa_{scaled,offset} = scale * (abscissa + offset)$. (Note: If no scale factor is specified via `X SCALE`, $scale = 1.0$.) This command is not available for analytic or piecewise analytic functions. `ABSCISSA OFFSET` is an alias for `X OFFSET`.
- The `Y SCALE` command line sets the scale factor value for the ordinate, which has the following effect: $ordinate_{scaled,offset} = scale * (ordinate + offset)$. (Note: If no offset is specified via `Y OFFSET`, $offset = 0.0$.) `ORDINATE SCALE` is an alias for `Y SCALE`.
- The `Y OFFSET` command line sets the offset value for the ordinate, which has the following effect: $ordinate_{scaled,offset} = scale * (ordinate + offset)$. (Note: If no scale factor is specified via `Y SCALE`, $scale = 1.0$.) `ORDINATE OFFSET` is an alias for `Y OFFSET`.
- The `ABSCISSA SCALE` command line is an alias for `X SCALE`.
- The `ABSCISSA OFFSET` command line is an alias for `X OFFSET`.
- The `ORDINATE SCALE` command line is an alias for `Y SCALE`.
- The `ORDINATE OFFSET` command line is an alias for `Y OFFSET`.
- The `DEBUG` command line prints functions to the log file if they were scaled and/or offset. This command line is optional and available for `PIECEWISE LINEAR` and `PIECEWISE CONSTANT` functions. The generated function name is the original function name concatenated with `_with_scale_and_offset_applied`. The generated function is a valid function and could be placed into the input file and used. The `DEBUG` command can also be used to print `ANALYTIC` and `PIECEWISE ANALYTIC` functions to the log file if they are evaluated using the command `EVALUATE FROM <real> TO <real> BY <real>`. The generated function name is the original function name concatenated with `_as_pieewise_linear`.
- The `VALUES` command block consists of the real value pairs (x_1, y_1) through (x_n, y_n), which describe the function. This command block must be used if the value on the `TYPE` command line is `CONSTANT`, `PIECEWISE LINEAR`, or `PIECEWISE CONSTANT`. For a `CONSTANT` function, only one value is needed. For a `PIECEWISE LINEAR` or `PIECEWISE CONSTANT` function, the values are (x, y) pairs of data that describe the function. The values are nested inside the `VALUES` command block.
- As an alternative to the `VALUES` command block, the `X` and `Y` data pair values can be read from an external file with the `DATA FILE` command. An example data file format is given below. All characters in a file line that follow the first '#' character in that line are considered as comments and ignored. The remainder of the file contains columns of Real numbers. The columns can be separated by white space, commas, or some combination of white space and commas. Optionally, the columns of data to represent `X` and `Y` can be specified by the `X FROM COLUMN <int>xCol Y FROM COLUMN <int>yCol` command. If no optional column number is specified, it is assumed that column 1 represents the `X` data and column 2 represents the `Y` data. The data file option is valid for piecewise linear and piecewise constant functions.

#-----

# Time	Pressure	Temperature
0.0	1.0e-3	1.5
0.1	1.2e-3	1.7
0.2	0.9e-3	2.9
0.3	0.1e-3	9.0

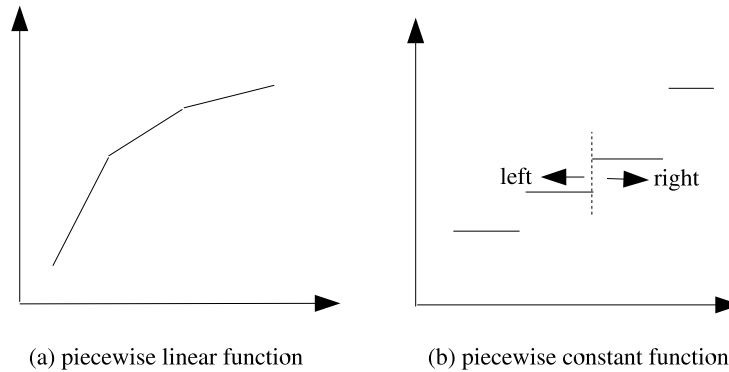


Figure 2.1: Piecewise linear and piecewise constant functions

- A `PIECEWISE LINEAR` function performs linear interpolations between the provided value pairs; a `PIECEWISE CONSTANT` function is constant valued between provided value pairs. Figure 2.1 (a) shows an example of a piecewise linear function, and Figure 2.1 (b) shows an example of a piecewise constant function.
- For functions that are based on tabular values, such as the `PIECEWISE LINEAR` and `PIECEWISE CONSTANT` functions, there is a possibility that the function may be evaluated for abscissa values that fall outside the range of the tabulated values. If the abscissa for which the function is to be evaluated is lower than the lowest tabulated abscissa, the function returns the ordinate corresponding to the smallest tabulated abscissa. Likewise, if the function is evaluated for an abscissa higher than the highest tabulated abscissa, the function returns the ordinate corresponding to the highest tabulated abscissa. For example, consider the following function:

```
begin function my_func
  type = piecewise linear
  begin values
    5.0  0.0
    10.0 50000.0
  end values
end function my_func
```

For values less than 5, this function returns 0, and for values greater than 10, it returns 50000.

- `PIECEWISE LINEAR` and `PIECEWISE CONSTANT` functions must be defined with either monotonically increasing x-values ($x_i \geq x_{i-1}$) or monotonically decreasing x-values ($x_i \leq x_{i-1}$). For example, the following function definition is invalid:

```

begin function my_func
  type = piecewise linear
  begin values
    0.0  0.0
    0.5  0.0
    1.0  1.0
    0.5  1.0
  end values
end function my_func

```

- For a piecewise constant function, a constant valued segment ends on the left hand side of an abscissa value and a new constant value segment begins on the right hand side of the same abscissa value. (This transition from one constant value to another is indicated by the dotted line in Figure 2.1 (b).) When the value of the function is to be evaluated at a discontinuity, where there two potential values for the ordinate, the default behavior is to use the ordinate from the value pair that has the lower-valued abscissa, or in other words, to use the value on the left hand side of the discontinuity. The `AT DISCONTINUITY EVALUATE TO` command line can be used to override this default behavior at an abscissa with two ordinate values. The command line can have a value of either `LEFT` or `RIGHT`. If `LEFT` (the default) is specified, the ordinate value to the left of the abscissa is used; if `RIGHT` is specified, the ordinate value to the right of the abscissa is used.
- The `PIECEWISE MULTIVARIATE` function is used with the Hydra Plasticity Material Model.
- The `MULTICOLUMN PIECEWISE LINEAR` function is not used in Sierra Solid Mechanics. It can be used in coupled analysis with Aria, Fuego, etc.
- The `EVALUATE EXPRESSION` command line consists of one or more user-supplied algebraic expressions. This command line must be used if the value on the `TYPE` command line is `ANALYTIC`. See the rules and options for composing algebraic expressions discussed below.
- The `DIFFERENTIATE EXPRESSION` command line consists of one or more user-supplied algebraic expressions. This command line must be used if the value on the `TYPE` command line is `ANALYTIC` and you need the derivative of the analytic function that you defined with `EVALUATE EXPRESSION`. This can happen if you are using an analytic function in a constitutive model. See the rules and options for composing algebraic expressions discussed below.

Importantly, a `FUNCTION` command block cannot contain both a `VALUES` command block and an `EVALUATE EXPRESSION` command line.

Rules and options for composing algebraic expressions. If you choose to use the `EVALUATE EXPRESSION` command line, you will need to write the algebraic expressions. The algebraic expressions are written using a C-like format. Each algebraic expression is terminated by a semi-colon(`;`). The entire set of algebraic expressions, whether a single expression or several, is enclosed in a single set of double quotes(`" "`).

An expression is evaluated with `x` as the independent variable. We first provide several simple examples and then list the options available in the algebraic expressions.

Example: Return $\sin(x)$ as the value of the function.

```
begin function sinx
  type is analytic
  evaluate expression is "sin(x)"
end function sinx
```

In this example, the table in comments is equivalent to the evaluated expression:

```
begin function pressure
  type is analytic
  evaluate expression is "      \#
    (x <= 0.0) ? (          \#
      0.0                \#
    ) : (                \#
      (x < 0.5) ? (        \#
        x*200.0            \#
      ) : (                \#
        100.0              \#
      )                    \#
    )                      \#
  "
  #      begin values
  #      0.0      0.0
  #      0.5      100.0
  #      1.0      100.0
  #      end values
end function pressure
```

The following functionality is currently implemented for the expressions:

Operators Valid arithmetic and Boolean operations are as follows:

Symbol	Operation
+, -	plus, minus
*, /	times, divide
^	power
==, !=	equivalent, not equivalent
>, <	greater/less than
>=, <=	greater/less than or equal to
!	not
&, , &&,	and, or
?, :	ternary if, then, else

Parentheses $()$, represents standard mathematical meaning of operation ordering.

Component indexing `value[i]`, accesses the i th component of a multi-component variable such as a vector, tensor, etc. This index is one-based. For a vector `vec`,

```

vec_x = vec[1]
vec_y = vec[2]
vec_z = vec[3]

```

for a symmetric tensor `sym`,

```

sym_xx = sym[1]
sym_yy = sym[2]
sym_zz = sym[3]
sym_xy = sym[4]
sym_yz = sym[5]
sym_zx = sym[6]

```

and for a full tensor `full`,

```

full_xx = full[1]
full_yy = full[2]
full_zz = full[3]
full_xy = full[4]
full_yz = full[5]
full_zx = full[6]
full_yx = full[7]
full_zy = full[8]
full_xz = full[9]

```

Math functions Valid mathematical operations are as follows:

Function	Operation
<code>abs(x)</code>	absolute value of x
<code>mod(x, y)</code>	modulus of $x y$
<code>min(x, y)</code>	minimum value of x, y
<code>max(x, y)</code>	maximum value of x, y
<code>sign(x)</code>	sign operator: -1 if x is negative, 1 if positive
<code>ipart(x)</code>	integer part of x
<code>fpart(x)</code>	fractional part of x

Power functions Valid functions for expressions containing exponents are as follows:

Function	Operation
<code>pow(x, y)</code>	x^y
<code>pow10(x)</code>	10^x
<code>sqrt(x)</code>	$\sqrt{x}$

Trigonometric functions Valid trigonometric operations are as follows:

Function	Operation
<code>acos(x)</code>	arccosine of x
<code>asin(x)</code>	arcsine of x
<code>asinh(x)</code>	inverse hyperbolic sin of x
<code>atan(x)</code>	arctangent of x
<code>atan2(y, x)</code>	arctangent of y/x , signs of x and y
<code>cos(x)</code>	cosine of x
<code>cosh(x)</code>	hyperbolic cosine of x
<code>sin(x)</code>	sine of x
<code>sinh(x)</code>	hyperbolic sine of x
<code>tan(x)</code>	tangent of x
<code>tanh(x)</code>	hyperbolic tangent of x

Logarithm functions Valid logarithmic operations are as follows:

Function	Operation
<code>log(x)</code> or <code>ln(x)</code>	natural logarithm of x
<code>log10(x)</code>	base-10 logarithm of x
<code>exp(x)</code>	e^x

Rounding functions Valid mathematical rounding operations are as follows:

Function	Operation
<code>ceil(x)</code>	smallest integer value not less than x
<code>floor(x)</code>	largest integer value not greater than x

Random functions Valid functions for generating random data are as follows:

Function	Operation
<code>random()</code>	random real number $0.0 \leq x < 1.0$
<code>random(x)</code>	seeds the random number generator
<code>time()</code>	elapsed time in seconds since January 1, 1970

Coordinate conversion functions Valid functions for converting to and from polar and Cartesian coordinate systems are as follows:

Function	Operation
<code>deg(x)</code>	converts x from radians to degrees
<code>rad(x)</code>	converts x from degrees to radians
<code>recttopolr(x, y)</code>	magnitude of vector (x, y)
<code>recttopola(x, y)</code>	angle of vector (x, y)
<code>poltorectx(r, th)</code>	x -coordinate of angle th at distance r
<code>poltorecty(r, th)</code>	y -coordinate of angle th at distance r

If, then, else syntax $A ? B : C$, where A is a Boolean statement such as $(x < 2.0)$, B is a statement (or set of statements) to be evaluated if A is true, and C is a statement (or set of statements) to be evaluated if A is false.

Other utilities Miscellaneous other utility functions are as follows:

- `cos_ramp(x, xstart, xend)` defines a cosine ramp loading function. For $x < xstart$, it returns 0.0. For $x > xend$, it returns 1.0. If $xstart \leq x \leq xend$, the function has the value $1.0 - \cos(\pi * (x - xstart) / (xend - xstart)) / 2.0$. The primary purpose for the cosine ramp is to provide smooth loading. If attempting to run a nearly quasistatic problem with the dynamic solver, a displacement boundary condition applied via the `cos_ramp` will generally give a smooth response for a given loading time.
- `cycloidal_ramp(x, xstart, xend)` defines a cycloidal ramp function. For $x < xstart$ it returns 0.0. For $x > xend$, it returns 1.0. If $xstart \leq x \leq xend$, the function has the value $(x - xstart) / (xend - xstart) - 1 / (2 * \pi) * \sin(2 * \pi / (xend - xstart) * (x - xstart))$. The primary purpose for the cycloidal front ramp is to provide smooth loading for prescribed displacements and velocities. This function has continuous acceleration derivatives at $xstart$ and $xend$, providing a smoother response than the `cos_ramp` for prescribed displacement boundary conditions.
- `haversine_pulse(x, xstart, xend)` defines a haversine pulse function. For $x < xstart$, it returns 0.0. For $x > xend$, it returns 0.0. If $xstart \leq x \leq xend$, the function has the value $\text{pow}(\sin(\pi * (x - xstart) / (xend - xstart)), 2)$. Design shock loads for components are often specified as haversine acceleration pulses; this function is included to make it easier to apply this loading.

Constants. There are three predefined constants that may be used in an expression. These three constants are `e`, `pi`, and `two_pi`. Note that these constants are reserved variable names and thus cannot be redefined in an expression.

Math symbol	Expression	Value
e	<code>e</code>	2.7182818284...
π	<code>pi</code>	3.1415926535...
2π	<code>two_pi</code>	$2 * 3.1415926535 \dots$

Also, there are two predefined constant functions that can be used, `SIERRA_CONSTANT_FUNCTION_ZERO` and `SIERRA_CONSTANT_FUNCTION_ONE`. These two functions are equivalent to defining a function with `TYPE = CONSTANT` with values of zero and one.

The `PIECEWISE ANALYTIC` function is a combination of several analytic functions that are evaluated over different ranges of x . As an example consider the following function. The function ramps up a load from time 0.0 to 1.0, holds that load constant from time 1.0 to 2.0, ramps the load back down to zero from time 2.0 to 3.0, and then holds the load constant again after time 3.0.

```

begin function force_ramp
  type is piecewise analytic
  begin expressions
    0.0 "(1.0-cos(pi*x))/2.0"
    1.0 "1.0"
    2.0 "(1.0+cos(pi*(x-2.0)))/2.0"
    3.0 "0.0"
  end
end

```

Note this function is equivalent to the single compound, ANALYTIC function below, though the piecewise version tends to be easier to read. The expression compound if then else statement below also demonstrates the recommended indentation for nested if statements. The statement could be written on a single line, but the indentation makes the logic of the evaluation statement more readable.

```

begin function force_ramp
  type is analytic
  evaluate expression = "
    (x < 1.0) ? (
      (1.0-cos(pi*x))/2.0
    ) : (
      (x < 2.0) ? (
        1.0
      ) : (
        (x < 3.0) ? (
          (1.0+cos(pi*(x-2.0)))/2.0
        ) : (
          0.0
        )
      )
    )
  "
end

```

2.1.6 General Functions

```

BEGIN FUNCTION <string>function_name
  TYPE = ANALYTIC|PIECEWISE ANALYTIC

  EXPRESSION VARIABLE: <var_name> = NODAL|NODAL_VECTOR|
    NODAL_TENSOR|NODAL_SYM_TENSOR|ELEMENT|ELEMENT_VECTOR|
    ELEMENT_TENSOR|ELEMENT_SYM_TENSOR|FACE|GLOBAL <var_source>

  BEGIN EXPRESSIONS
    <real>x_1    <string>analytic_expression_1
    <real>x_2    <string>analytic_expression_2
    ...

```



```

    <real>x_n    <string>analytic_expression_n
END
AT DISCONTINUITY EVALUATE TO <string>LEFT|RIGHT (LEFT)
EVALUATE EXPRESSION = <string>analytic_expression1;
    analytic_expression2; ...
END [FUNCTION <string>function_name]

```

General functions can compute a global, element, or nodal scalar value that is a function of several input values. Each input value to the function is explicitly defined via an `EXPRESSION VARIABLE` command. The expression variable command defines variable for use in an expression or expression list. As an example consider a function to apply a combination of shearing and extension to a block. An example displacement map going from time zero to time one is shown in Figure 2.2. The equation for this displacement map is given by:

$$\Delta_x(x, y, t) = t * (0.5 * x + 0.5 * y) \quad (2.1)$$

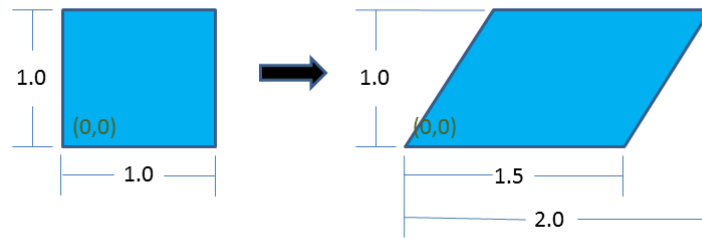


Figure 2.2: Example Displacement Map

This displacement map can be applied by a combination of a general function and a displacement boundary condition by:

```

BEGIN FUNCTION delta_x
  TYPE = analytic
  EXPRESSION variable: mx    = NODAL model_coordinates(x)
  EXPRESSION variable: my    = NODAL model_coordinates(y)
  EXPRESSION variable: time  = GLOBAL time
  EVALUATE EXPRESSION = "time * (0.5*mx + 0.5*my) "
END

BEGIN PRESCRIBED DISPLACEMENT
  BLOCK = BLOCK_1
  COMPONENT = X
  FUNCTION = delta_x
END

```

The `model_coordinates` are the initial nodal coordinates. Time is the current analysis time. If a general function is being applied to a nodal boundary condition, such as displacement, function

will be evaluated at each node. On a node the value of `model_coordinates` will be extracted and placed into the expression variables 'mx' and 'my'. The analysis time is placed into the expression variable 'time'. The expression is then evaluated producing a single value at the node, the nodal value is used to define the x displacement at that particular node.

A function used to define nodal quantities may reference NODAL or GLOBAL expression variables. A function used to define an element quantity may reference ELEMENT or GLOBAL expression variables. A function used to define a global quantity may only reference GLOBAL expression variables. Generally any quantity that would be available for output at nodes, at elements, or as a global variable may be used as an expression variable.

Expression variables may also accept vectors, tensors or symmetric tensors. A vector can define variables in terms of x, y, and z. For example,

```
EXPRESSION variable: A = element_vector velocity
```

would define `A_x`, `A_y`, and `A_z` for use in an expression. A symmetric tensor can define variables in terms of xx, yy, zz, zy, yz, and zx. For example,

```
EXPRESSION variable: A = element_sym_tensor unrotated_stress
```

would define `A_xx`, `A_yy`, `A_zz`, `A_xy`, `A_yz`, and `A_zx` for use in an expression. A tensor can define variables in terms of xx, yy, zz, zy, yz, zx, yz, xz, and zy.

```
EXPRESSION variable: A = element_tensor rotation
```

would define `A_xx`, `A_yy`, `A_zz`, `A_xy`, `A_yz`, `A_zx`, `A_yx`, `A_zy`, and `A_xz`. Particularly for tensors, this provides a clearer syntax for accessing the components of a multi-component variable.

Example of use:

```
# Define using named vector components
BEGIN FUNCTION velocity_vec
  TYPE = analytic
  EXPRESSION variable: V = nodal_vector velocity
  EVALUATE EXPRESSION = "sqrt((V_x)^2 + (V_y)^2 + (V_z)^2 )"
END

# Equivalent function using general variable indexing
BEGIN FUNCTION velocity_index
  TYPE = analytic
  EXPRESSION variable: V = nodal velocity
  EVALUATE EXPRESSION = "sqrt(V[1]^2 + V[2]^2 + V[3]^2 )"
END

...

BEGIN USER OUTPUT
  # speed1 and speed2 are equivalent, both are a nodal field that
  # is the the local magnitude of the nodal velocity field
```

```

compute nodal speed1 as function velocity_vec
compute nodal speed2 as function velocity_index
END

```

2.1.7 Axes, Directions, and Points

```

DEFINE POINT <string>point_name WITH COORDINATES
    <real>value_1 <real>value_2 <real>value_3
DEFINE DIRECTION <string>direction_name WITH VECTOR
    <real>value_1 <real>value_2 <real>value_3
DEFINE AXIS <string>axis_name WITH POINT
    <string>point_1 POINT <string>point_2
DEFINE AXIS <string>axis_name WITH POINT
    <string>point DIRECTION <string>direction

```

Many Sierra/SM features require the definition of geometric entities. For instance, the prescribed displacement boundary condition requires a direction definition, and the cylindrical velocity initial condition requires an axis definition. Currently, Sierra/SM input permits the definition of points, directions, and axes. Definition of these geometric entities occurs in the SIERRA scope.

The `DEFINE POINT` command line is used to define a point:

```

DEFINE POINT <string>point_name WITH COORDINATES
    <real>value_1 <real>value_2 <real>value_3

```

where

- The string `point_name` is a name for this point. This name must be unique to all other points defined in the input file.
- The real values `value_1`, `value_2`, and `value_3` are the x , y , and z coordinates of the point.

The `DEFINE DIRECTION` command line is used to define a direction:

```

DEFINE DIRECTION <string>direction_name WITH VECTOR
    <real>value_1 <real>value_2 <real>value_3

```

where

- The string `direction_name` is a name for this direction. This name must be unique to all other directions defined in the input file.
- The real values `value_1`, `value_2`, and `value_3` are the x , y , and z magnitudes of the direction vector.

There are two command lines that can be used to define an axis. The first `DEFINE AXIS` command line uses two points:

```

DEFINE AXIS <string>axis_name WITH POINT
    <string>point_1 POINT <string>point_2

```

where

- The string `axis_name` is a name for this axis. This name must be unique to all other axes defined in the input file.
- The strings `point_1` and `point_2` are the names for two points defined in the input file via a `DEFINE POINT` command line.

The second `DEFINE AXIS` command line uses a point and a direction:

```
DEFINE AXIS <string>axis_name WITH POINT
    <string>point DIRECTION <string>direction
```

where

- The string `axis_name` is a name for this axis. This name must be unique to all other axes defined in the input file.
- The string `point` is the name of a point defined in the input file via a `DEFINE POINT` command line.
- The string `direction` is the name of a direction defined in the input file via a `DEFINE DIRECTION` command line.

2.1.8 Orientation

```
BEGIN ORIENTATION <string>orientation_name
    SYSTEM = <string>RECTANGULAR|Z RECTANGULAR|CYLINDRICAL|
    SPHERICAL(RECTANGULAR)
    #
    POINT A = <real>global_ax <real>global_ay <real>global_az
    POINT B = <real>global_bx <real>global_by <real>global_bz
    #
    ROTATION ABOUT <integer> 1|2|3(2) = <real>theta(0.0)
END [ORIENTATION <string>orientation_name]
```

The `ORIENTATION` command block is currently used in Sierra/SM to define a local *co-rotational* coordinate system at each shell element centroid for output of shell in-plane stresses and strains. Each `BEGIN SHELL SECTION` command block has an orientation that generates this local coordinate system for each element in the associated element blocks. The generated local coordinate system rotates with the shell element, thus making it *co-rotational*.

Stresses and strains for shell elements are computed in the shell element's local coordinate system, which typically varies from one element to the next, and can be seen in Figure 2.3. The local **R** axis of element 1 does not align with the local **R** axis of element 2. Thus without the use of an orientation to rotate the local coordinate system in to a consistent coordinate system, stress and strain results would be difficult to decipher. The use of an orientation generates a transformation matrix from a shell element's local **R**, **S**, **T** coordinate system to the final coordinate system, **R'**, **S'**, **T'**, by use of orientation and element-specific basis vectors and a rotation about an axis, all defined below.

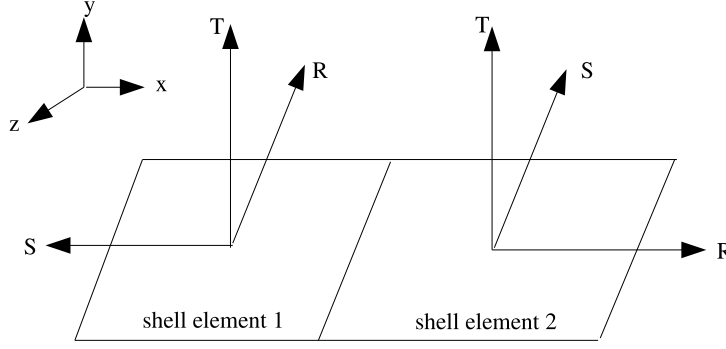


Figure 2.3: Adjacent shell elements with non-aligned local coordinate systems

There are four orientation systems available in Sierra/SM to better align the stress and strain output of shell elements. These orientations are defined via the optional `SYSTEM` command line within the `ORIENTATION` command block and are: `RECTANGULAR`, `Z_RECTANGULAR`, `CYLINDRICAL` and `SPHERICAL` with the default being `RECTANGULAR`. If the `ORIENTATION` command line within a `SHELL SECTION` block is not used, the default orientation is a `RECTANGULAR` system with axes aligned with the global X,Y and Z axes.

Each of the four systems produces a unique set of bases vectors, $\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3$, computed using the centroid of the shell element and the two required inputs, `POINT A` and `POINT B`. These four methods of computing $\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3$ are described below.

- **RECTANGULAR:** `POINT A` defines a point, P_A , in the local coordinate system of the shell element that lies in the direction of the first basis vector, $\mathbf{g}_1$, from the centroid, C with local coordinate of $(0, 0, 0)$, see Figure 2.4. Thus, the vector from the centroid of the element to point P_A is defined as $\mathbf{p}_A = P_A - C$. Similarly, a vector from the centroid to point P_B , defined via `POINT B`, can be computed as $\mathbf{p}_B = P_B - C$. Using these two vectors, the rectangular basis vectors are defined as: $\mathbf{g}_1 = \frac{\mathbf{p}_A}{\|\mathbf{p}_A\|}$, $\mathbf{g}_3 = \frac{\mathbf{g}_1 \times \mathbf{p}_B}{\|\mathbf{g}_1 \times \mathbf{p}_B\|}$, $\mathbf{g}_2 = \frac{\mathbf{g}_3 \times \mathbf{g}_1}{\|\mathbf{g}_3 \times \mathbf{g}_1\|}$.
- **Z_RECTANGULAR:** This set of basis vectors is defined similarly to the `RECTANGULAR` system, however point P_A now defines a point that lies in the direction of the third basis vector, $\mathbf{g}_3$ from the centroid of the element, see Figure 2.5. Using the same definition for $\mathbf{p}_A$ and $\mathbf{p}_B$ as above we can define the three basis vectors as: $\mathbf{g}_3 = \frac{\mathbf{p}_A}{\|\mathbf{p}_A\|}$, $\mathbf{g}_2 = \frac{\mathbf{g}_3 \times \mathbf{p}_B}{\|\mathbf{g}_3 \times \mathbf{p}_B\|}$, $\mathbf{g}_1 = \frac{\mathbf{g}_2 \times \mathbf{g}_3}{\|\mathbf{g}_2 \times \mathbf{g}_3\|}$.
- **CYLINDRICAL:** `POINT A`, P_A , and `POINT B`, P_B , define the direction of the third basis vector, $\mathbf{g}_3$ such that $\mathbf{g}_3 = \frac{P_B - P_A}{\|P_B - P_A\|}$, as seen in Figure 2.6. Defining a vector, $\mathbf{v}$, from P_A to the centroid of the element, C in the global coordinate system, the second basis vector, $\mathbf{g}_2$ is defined as: $\mathbf{g}_2 = \frac{\mathbf{v} \times \mathbf{g}_3}{\|\mathbf{v} \times \mathbf{g}_3\|}$. Therefore the first basis vector is defined as $\mathbf{g}_1 = \frac{\mathbf{g}_2 \times \mathbf{g}_3}{\|\mathbf{g}_2 \times \mathbf{g}_3\|}$.
- **SPHERICAL:** The point P_A , from the `POINT A` command line, defines the center of a sphere. The point P_B , from the `POINT B` command line, defines a polar axis for the sphere. (See Figure 2.7.) The first basis vector is defined by the vector that passes from P_A to the centroid, C in the global coordinate system, of the element: $\mathbf{g}_1 = \frac{C - P_A}{\|C - P_A\|}$. The third basis vector is parallel to the vector from P_A to P_B such that $\mathbf{g}_3 = \frac{P_B - P_A}{\|P_B - P_A\|}$. From these two vector, the

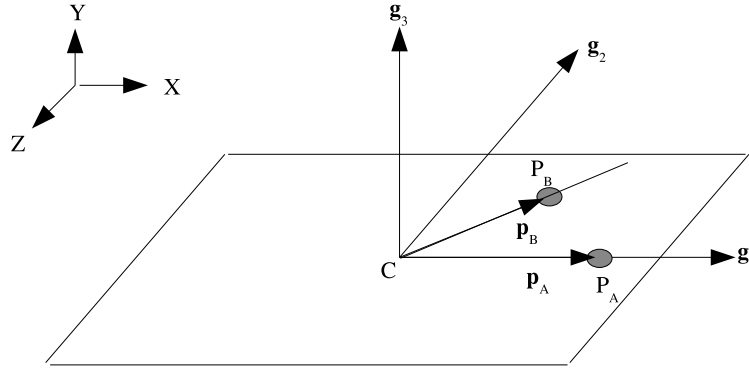


Figure 2.4: Rectangular coordinate system

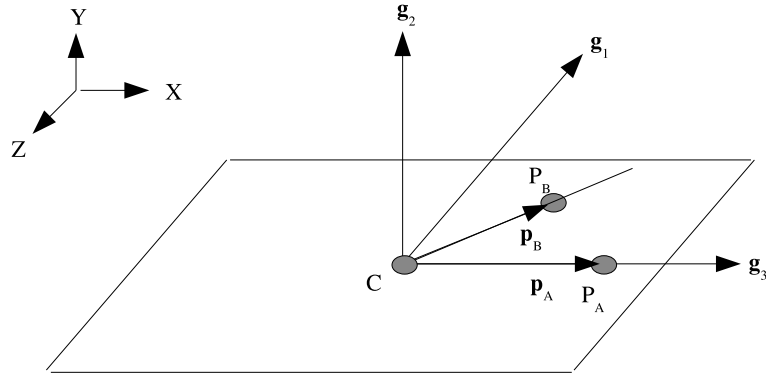


Figure 2.5: Z-Rectangular coordinate system.

second basis vector is defined as $\mathbf{g}_2 = \frac{\mathbf{g}_3 \times \mathbf{g}_1}{\|\mathbf{g}_3 \times \mathbf{g}_1\|}$. If $\mathbf{g}_1$ and $\mathbf{g}_3$ are collinear, the second basis vector is taken to be the normal of the element, $\mathbf{T}$, and the third basis is $\mathbf{g}_3 = \frac{\mathbf{g}_1 \times \mathbf{g}_2}{\|\mathbf{g}_1 \times \mathbf{g}_2\|}$.

It is possible, using the `ROTATION ABOUT` command line, to specify an angle and an axis to rotate the basis vectors, $\mathbf{g}_1$, $\mathbf{g}_2$ and $\mathbf{g}_3$ about in order to produce a second set of basis vectors, $\mathbf{g}'_1$, $\mathbf{g}'_2$ and $\mathbf{g}'_3$, that are then used to compute the final transformation matrix. The syntax for this command is as follows: `ROTATION ABOUT 1|2|3(2) = <real>theta(0.0)`

Where the 1,2,3 refers to which basis vector the rotation will occur about. The default value is 2 with an angle theta of 0.0, thus implying a rotation of 0.0 radians about $\mathbf{g}_2$. How $\mathbf{g}'_1$, $\mathbf{g}'_2$ and $\mathbf{g}'_3$ are computed for each of the three cases is described below. In each case, the second order rotation

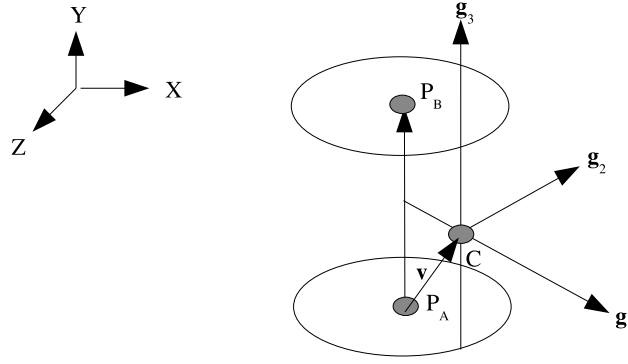


Figure 2.6: Cylindrical coordinate system.

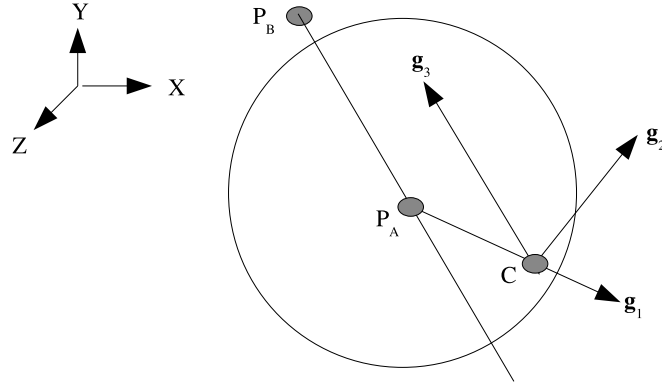


Figure 2.7: Spherical coordinate system.

tensor, $\mathbf{Q}$, is defined as:

$$\mathbf{Q} = \begin{bmatrix} 2(q_0^2 + q_1^2) - 1 & 2(q_1q_2 - q_0q_3) & 2(q_1q_3 - q_0q_2) \\ 2(q_1q_2 + q_0q_3) & 2(q_0^2 + q_2^2) - 1 & 2(q_2q_3 - q_0q_1) \\ 2(q_1q_3 - q_0q_2) & 2(q_2q_3 + q_0q_1) & 2(q_0^2 + q_3^2) - 1 \end{bmatrix}$$

Where: $q_0 = \cos(\frac{\theta}{2})$, $q_1 = a_1 \sin(\frac{\theta}{2})$, $q_2 = a_2 \sin(\frac{\theta}{2})$, and $q_3 = a_3 \sin(\frac{\theta}{2})$, where the vector $\mathbf{a} = [a_1, a_2, a_3]$ is the basis vector we are rotating about.

- Rotation about $\mathbf{g}_1$ (ROTATION ABOUT 1): The prime set of basis vectors are defined as: $\mathbf{g}'_1 = \mathbf{Q}\mathbf{g}_3$, $\mathbf{g}'_2 = \mathbf{Q}\mathbf{g}_2$, $\mathbf{g}'_3 = -\mathbf{g}_1$. Which can graphically be seen in Figure 2.8
- Rotation about $\mathbf{g}_2$ (ROTATION ABOUT 2), default: The prime set of basis vectors are defined as: $\mathbf{g}'_1 = \mathbf{Q}\mathbf{g}_1$, $\mathbf{g}'_2 = \mathbf{Q}\mathbf{g}_3$, $\mathbf{g}'_3 = -\mathbf{g}_2$.
- Rotation about $\mathbf{g}_3$ (ROTATION ABOUT 3): The prime set of basis vectors are defined as: $\mathbf{g}'_1 = \mathbf{Q}\mathbf{g}_2$, $\mathbf{g}'_2 = \mathbf{Q}\mathbf{g}_1$, $\mathbf{g}'_3 = -\mathbf{g}_3$.

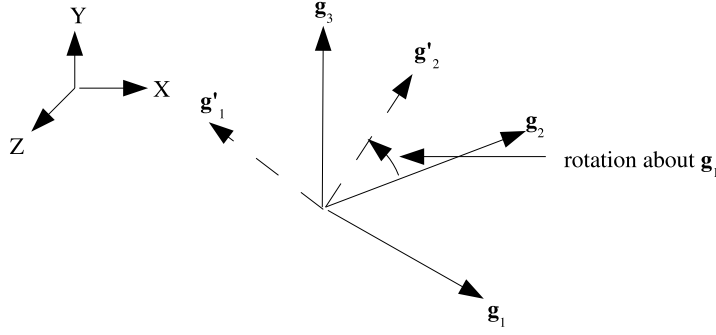


Figure 2.8: Rotation about 1

The next step is to project $\mathbf{g}'_1$ onto the shell face as follows: $\mathbf{R}' = \mathbf{g}'_1 - (\mathbf{g}'_1 \cdot \mathbf{T})\mathbf{T}$, which defines a rotated $\mathbf{R}$ vector. If $\mathbf{R}'$ is perpendicular to the face of the element, we then redefine $\mathbf{R}'$ to be: $\mathbf{R}' = \mathbf{g}'_2 - (\mathbf{g}'_2 \cdot \mathbf{T})\mathbf{T}$, the projection of $\mathbf{g}'_2$ onto the element face. Once we have $\mathbf{R}'$ we can define $\mathbf{S}'$ as $\frac{\mathbf{T} \times \mathbf{R}'}{\|\mathbf{T} \times \mathbf{R}'\|}$. Hence, the final transformation matrix taking local stress and strain (in the $\mathbf{R}, \mathbf{S}$ coordinate system) to the $\mathbf{R}', \mathbf{S}'$ coordinate system is:

$$\mathbf{Q}' = \begin{bmatrix} \mathbf{R} \cdot \mathbf{R}' & \mathbf{R} \cdot \mathbf{S}' & 0 \\ \mathbf{S} \cdot \mathbf{R}' & \mathbf{S} \cdot \mathbf{S}' & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

2.1.9 Coordinate System Block Command

```
BEGIN COORDINATE SYSTEM <string>coordinate_system_name
  TYPE = <string>RECTANGULAR|CYLINDRICAL|SPHERICAL (RECTANGULAR)
  #
  ORIGIN = <string>origin_point_name
  VECTOR = <string>z_vector_point_name
  POINT = <string>x_vector_point_name
  ORIGIN NODE = <string>origin_nodelist_name
  VECTOR NODE = <string>z_vector_nodelist_name
  POINT NODE = <string> x_vector_nodelist_name
  #
END [COORDINATE SYSTEM <string>coordinate_system_name]
```

The `COORDINATE SYSTEM` command block is used to define a local coordinate system for transforming a stress tensor from components in the global x, y, z coordinate system to the local system for output. This command block cannot be used for in-plane stresses and strains of shell elements because the shell integration point stresses and strains are computed in a local element system that varies element to element and rotates with the element. For output of shell stresses, the `BEGIN ORIENTATION` command must be used as explained in the previous section.

The `COORDINATE SYSTEM` block is also used to define a local coordinate system on the elements of a block for use in a representative volume analysis. In these analyses, elements of each representative volume will be aligned with the local system defined on the parent element.

A local element coordinate system may be defined using points defined in the input file with the `DEFINE POINT` command or using node lists in the mesh file that contain exactly one node each. By using nodes in the mesh file, the defined coordinate system may translate and rotate during the analysis.

The `TYPE` command line allows three options for constructing a local coordinate system: `RECTANGULAR`, `CYLINDRICAL`, and `SPHERICAL`. The type defaults to `RECTANGULAR` if the `TYPE` command line is not present.

- **RECTANGULAR:** The command line `ORIGIN` or `ORIGIN NODE` specifies the origin of the local Cartesian coordinate system. The local Z' axis is a vector from the origin to the point given in the `VECTOR` or `VECTOR NODE` command. The local X' axis is the component of the vector from the origin to the `POINT` or `POINT NODE` that is orthogonal to the Z' axis. Finally, the local Y' axis is obtained from the cross product of Z' and X' .
- **CYLINDRICAL:** The command line `ORIGIN` or `ORIGIN NODE` specifies one point on the axis of the cylinder. The local Z' axis, the axis of the cylinder, is a vector from the origin to the point given in the `VECTOR` or `VECTOR NODE` command. The local X' axis is constructed as the vector normal to the cylindrical axis and passing through the location at which the local system is desired. Depending on the context, this local point may be a node, an element centroid, or an element integration point. If this local point lies on the Z' axis, then the point defined by the `POINT` or `POINT NODE` command is instead used to define the X' axis. Finally, the Y' axis is obtained from the cross product of Z' and X' . Thus at the desired point not on the cylinder axis, the X' axis is through the cylinder thickness, Y' is tangent to the cylinder, and Z' is parallel to the cylinder axis.
- **SPHERICAL:** The command line `ORIGIN` or `ORIGIN NODE` specifies the location of the center of the sphere. The local X' axis is constructed as the vector from the center through the location at which the local system is desired. Depending on the context, this local point may be a node, an element centroid, or an element integration point. The local Z' axis is the component of the vector from the origin to the point given in the `VECTOR` or `VECTOR NODE` command that is normal to the X' axis. If this Z' is parallel the X' axis, then the Z' axis is defined along the vector from the point defined in `POINT` or `POINT NODE` to the origin. Finally, the Y' axis is obtained from the cross product of Z' and X' . Thus at a desired point, the X' axis is through the sphere thickness and the Y' and Z' axes lie in the tangent plane.

2.1.10 Define Coordinate System Line Command

```
DEFINE COORDINATE SYSTEM <string>coord_sys_name
    <string>coord_sys_type WITH POINT <string>point_1
    POINT <string>point_2 POINT <string>point_3
```

The line command `DEFINE COORDINATE SYSTEM` can also be used to define the axis directions of a local coordinate system to be located at nodes, element centroids, or element integration points. In this command

- The string `coord_sys_name` is a name for this coordinate system. This name must be unique to all other coordinate systems defined in the input file.
- The string `coord_sys_type` states the type of the coordinate system to be used. Three options are allowed for constructing a local coordinate system: `RECTANGULAR`, `CYLINDRICAL`, and `SPHERICAL`.
- The strings `point_1`, `point_2`, and `point_3` are the names for three points defined in the input file via `DEFINE POINT` command lines. These three points are used to define two of the coordinate system axes as described below for the different types of systems.

For a rectangular system, the local Z' axis is parallel to the vector from the `point_1` to `point_2`. The local X' axis is the component of the vector from `point_1` to `point_3` that is orthogonal to the Z' axis. Finally, the local Y' axis is obtained from the cross product of Z' and X' .

Likewise, for a cylindrical system, the local Z' axis is parallel to the vector from the `point_1` to `point_2` and defines the axis of the cylinder. The local X' axis is constructed as the vector normal to the cylindrical axis and passing through the location at which the local system is desired. This local point may be a node, an element centroid, or an element integration point. If this local point lies on the Z' axis, then the point `point_3` is instead used to define the X' axis. Finally, the Y' axis is obtained from the cross product of Z' and X' . Thus at the desired point not on the cylinder axis, the X' axis is through the cylinder thickness, Y' is tangent to the cylinder, and Z' is parallel to the cylinder axis.

For a spherical system, `point_1` specifies the center of a sphere. The local X' axis is constructed as the vector from `point_1` through the location at which the local system is desired. This local point may be a node, an element centroid, or an element integration point. The local Z' axis is the component of the vector from the `point_1` to `point_3` that is normal to the X' axis. If this Z' is parallel the X' axis, then `point_3` is used instead. Finally, the Y' axis is obtained from the cross product of Z' and X' . Thus at a desired point, the X' axis is through the sphere thickness and the Y' and Z' axes lie in the tangent plane.

2.2 Procedure and Region

Explicit dynamics analyses use a Presto procedure and region. The Presto procedure scope is nested within the SIERRA scope, and the Presto region scope is nested within the procedure scope (see Section 1.3 for more information about scope). To create the scopes for the Presto procedure and Presto region for an explicit dynamics analysis, use the following commands:

```
BEGIN PRESTO PROCEDURE <string>procedure_name

    PRINT BANNER INTERVAL = <integer>print_banner_interval(MAX_INT)
    STOP AFTER INITIAL WRITE

    #
    # TIME CONTROL command block
    #
    BEGIN PRESTO REGION <string>region_name
        #
        # command blocks and command lines that appear in the
        # region scope
        #
    END [PRESTO REGION <string>region_name]
END [PRESTO PROCEDURE <string>procedure_name]
```

The `PRINT BANNER INTERVAL` line command sets the number of output lines to print before re-printing the column headings. The default value is set to the maximum allowed integer value. This means that the banner will be printed to the log file one time by default.

The `STOP AFTER INITIAL WRITE` line command ends the procedure after initialization. This tool can be useful in an analysis with restart.

Implicit analyses follow a similar structure, but use an Adagio procedure and region. The following example shows the structure of an input file for an implicit analysis:

```
BEGIN ADAGIO PROCEDURE <string>procedure_name
    #
    # TIME CONTROL command block
    #
    BEGIN ADAGIO REGION <string>region_name
        #
        # command blocks and command lines that appear in the
        # region scope
        #
    END [ADAGIO REGION <string>region_name]
END [ADAGIO PROCEDURE <string>region_name]
```

For both implicit and explicit analyses, the `TIME CONTROL` command block appears within the `PRESTO PROCEDURE` or `ADAGIO PROCEDURE` command block but outside of the `PRESTO`

REGION or ADAGIO REGION command block. These three command blocks (procedure, time control, and region) are discussed below.

Many command blocks and command lines fall within the region scope. These command blocks and command lines are described in other sections of this document.

2.2.1 Procedure

The analysis time, from the initial time to the termination time, is controlled within the procedure scope defined by the PRESTO PROCEDURE or ADAGIO PROCEDURE command block. For explicit dynamic analyses, the command block begins with an input line of the form:



```
BEGIN PRESTO PROCEDURE <string>procedure_name
```

and is terminated with an input line of the following form:

```
END [PRESTO PROCEDURE <string>procedure_name]
```

Implicit analyses use an Adagio procedure, which starts with the following line:



```
BEGIN ADAGIO PROCEDURE <string>procedure_name
```

and ends with the following line:

```
END [ADAGIO PROCEDURE <string>procedure_name]
```

The string `procedure_name` is the name for the procedure.

2.2.2 Time Control

Within the procedure scope, there is a TIME CONTROL command block. This command block lets the user set the initial time and the termination time for an analysis. This block also allows the user to control the size of the time step. The time step is controlled in different ways for explicit and implicit analyses, as explained below.

2.2.2.1 Explicit Dynamic Time Control

When Sierra/SM is used to run explicit transient dynamics analyses, it must run at a time step that is less than the critical time for the problem at any given instant. Typically, this global critical time step is based on a critical time step estimate calculated for each element. With the TIME CONTROL command block, the user can set an initial time step, scale the element-based time step estimate, and control the growth of the element-based estimate for the critical time step.

In addition to the element-based method for estimating the critical time step, Sierra/SM offers other methods for estimating the critical time step. One approach for estimating the critical time step is to calculate the maximum eigenvalue for the model. Two methods are implemented for calculating the maximum eigenvalue: the Lanczos method and the power method. A second approach for estimating the critical time step is to use a node-based method. The command blocks for implementing these various methods (maximum eigenvalue calculation and node-based) are in the region scope. There is also a mass-scaling technique that will influence the magnitude of the critical time step. If you use the mass-scaling technique, you must use the node-based method to

obtain a critical time step estimate.

The estimation of the time step is a key part of any explicit Sierra/SM analysis. Time step determination and control is discussed in detail in Chapter 3 of this document. The `TIME CONTROL` command block with its associated command lines are described in detail in Chapter 3. Consult Chapter 3 to determine how to specify command lines associated with the `TIME CONTROL` command block and how the `TIME CONTROL` command block fits into the overall scheme for time step control in explicit transient dynamics. Also see Chapter 3 to learn about the other methods for estimating the critical time step and the mass-scaling technique.

2.2.2.2 Implicit Time Control

When running implicit dynamic or quasistatic analyses with Sierra/SM a solver is used to find a converged solution at every time step. The time steps can be arbitrarily large, although errors due to time discretization are larger with larger time steps. In addition, for nonlinear problems, it is often more difficult to obtain a converged solution with large time steps. The time step size is controlled by the user, although Sierra/SM has a capability to optionally modify the time step based on the effort required by the nonlinear solver.

The `TIME CONTROL` block contains the basic commands used to control the time step size. To use adaptive time stepping, an `ADAPTIVE TIME STEPPING` command block is placed in the Adagio region scope. In addition, for all implicit analyses, commands to control the solver are required in the Adagio region scope. The details of the commands related to time stepping and the nonlinear solver are documented in Chapter 4.

2.2.3 Region

Individual time steps are controlled within the region scope. For explicit analyses, the region scope is defined by a `PRESTO REGION` command block that begins with an input line of the form

```
BEGIN PRESTO REGION <string>region_name
```

and is terminated with an input line of the following form:

```
END [PRESTO REGION <string>region_name]
```

Implicit analyses define the region scope using a `ADAGIO REGION` command block that begins with the following line:

```
BEGIN ADAGIO REGION <string>region_name
```

and is terminated by the following line:

```
END [ADAGIO REGION <string>region_name]
```

The string `region_name` is the name for this region.

The region, as indicated previously, determines what happens at each time step. In the procedure, we set the begin time and end time for the analysis. Time is incremented in the region. It is in the region where we set information about what occurs at various time steps. The output of results, for example, is set by command blocks in the region. If we want results output at certain times or



certain steps in the analysis, this information is set in command blocks in the region. The region also contains command blocks for the boundary conditions. A boundary condition can have a time-varying component. The region determines the value of the component for the current time step.

Two of the major types of command blocks, those for results output and boundary conditions, have already been mentioned. Other major types of command blocks in the region are those for restart control and contact. The region is also where the user selects the analysis model (finite element mesh). For implicit analyses, the command blocks for the solver are in the region. These blocks are discussed in Chapter 4.

The region makes use of information in the procedure and the SIERRA scope. For example, the specific element type used for an element block in the analysis model is defined in the SIERRA scope. This information about the element type is collected into an analysis model. The region then references this analysis model. As another example, the boundary condition command blocks can reference a function. The function will be defined in the SIERRA scope.

2.3 Use Finite Element Model

```
USE FINITE ELEMENT MODEL <string>model_name
```

The model specification occurs within the region scope. To specify the model (finite element mesh), use this command line. The string `model_name` must match a name used in a `FINITE ELEMENT MODEL` command block described in Section 6.1. If one of these command blocks uses the name `penetrator` in the command-block line and this is the model we wish to use in the region scope, then we would enter the command line as follows:

```
USE FINITE ELEMENT MODEL penetrator
```

2.4 Mesh Quality

```
BEGIN MESH QUALITY
  CHECK MESH TOPOLOGY = <string>ON|OFF(ON)
END [MESH QUALITY]
```

The `BEGIN MESH QUALITY` command block occurs within the region scope. This block will eventually offer a variety of options for diagnosing the quality and validity of the input mesh. Currently, the only option is `CHECK MESH TOPOLOGY`.

The `CHECK MESH TOPOLOGY` option checks for valid solid element connectivity and topology. In particular, it checks if any 3D bulk element is repeated twice in the mesh and if there are multiple elements which are (un-physically) attached to the same side of one face.

This check is on by default, any errors found trigger a fatal error and abort the run. The check can be manually turned off by including the command block with the `OFF` option.

2.5 Element Distortion Metrics

Sierra/SM can compute many element distortion metrics useful for assessing the quality of the solution as the mesh evolves over time. These metrics are computed as element variables, and can be used for output or as criteria for element death, just like any other element variable. The solution quality generally deteriorates as elements approach inversion, and if they do invert, the analysis aborts. The distortion metrics measure how close an element is to inversion. For all metrics a value of 1.0 is an ideal element and a value of 0.0 is a degenerate element. The following distortion metrics are available:

- `NODAL_JACOBIAN_RATIO` is currently available only for 8 node hexahedra and four node quads. This metric evaluates the Jacobian function at each node of each element, and then computes the nodal Jacobian ratio as the smallest nodal Jacobian divided by the largest nodal Jacobian. An element shaped like a parallelepiped with equal angles between all adjacent edges has a nodal Jacobian ratio of 1.0. A negative nodal Jacobian ratio indicates that the element is becoming either concave or locally inverted. See Figure 2.9 for examples of quadrilateral elements with positive, zero, and negative nodal Jacobian ratios. The element calculations on poorly shaped elements (those with negative nodal Jacobian ratios) will generally be less accurate than those on well shaped elements (those with positive nodal Jacobian ratios). In addition, contact surfaces may become tangled and non-physical if elements start to invert.

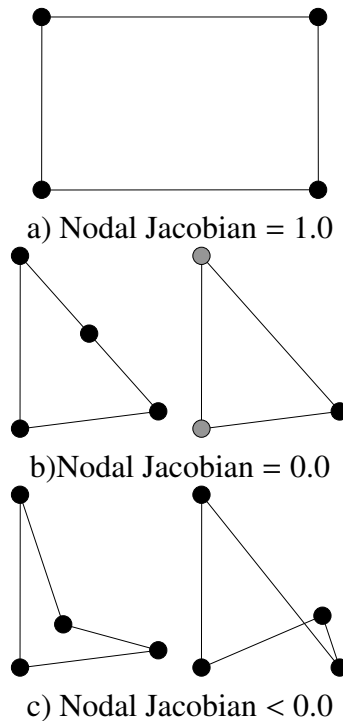


Figure 2.9: Examples of elements with varying nodal Jacobians

- `SCALED_JACOBIAN` is currently available only for 8 node hexahedra. This is the minimum value of the nodal jacobian normalized by the element volume. An element with right angle

edges will have a scaled jacobian of 1.0. A value of 0.0 indicates concavity or inversion. The `SCALED_JACOBIAN` is equivalent to the 'scaled jacobian' metric reported by cubit.

- `ELEMENT_SHAPE` computes a general shape metric for a variety of element topologies. Is currently defined for 8 node hexahedra, 4 node tetrahedra, 4 node quadrilaterals, and 3 node triangles. A value of 1.0 is optimal. A value of 0.0 represents a degenerate element. A negative value represents either a severely warped element or inverted element. For the 8 node hexahedron, the element shape is a scaled Jacobian equivalent to the 'shape' metric reported by cubit. For a tetrahedron or triangle, the shape is proportional to the element aspect ratio. For the 4 node quad element, the shape is proportional to the amount of warping of the quad. If a mesh contains elements with negative `ELEMENT_SHAPE`, problems could arise in the material or contact computations.
- `ASPECT_RATIO` is available only for tetrahedral elements. A perfect equilateral tetrahedron has an aspect ratio of 1.0. A degenerate zero-volume tetrahedron has an aspect ratio of zero. An inverted tetrahedron has a negative aspect ratio. A really thin element can have a extremely small aspect ratio.
- `SOLID_ANGLE` computes the minimum or maximum angle between edges of an element as compared to optimal angles. The optimal solid angle for tetrahedra and triangles is 60 degrees; for hexahedra and quadrilaterals, it is 90 degrees. This error metric is 1.0 for an element in which all angles are optimal. Severely distorted or twisted elements have values of this metric near 0.0, and it is negative for inverted elements. This metric is 0 for degenerate elements. The `SOLID_ANGLE` metric functions with hexahedron, tetrahedron, triangle, and quadrilateral elements.
- `PERIMETER_RATIO` measures the ratio of the deformed perimeter of an element to the undeformed perimeter of the element. This metric initially has a value of 1.0, and assumes values either greater than or lower than 1 as the mesh deforms. The `PERIMETER_RATIO` metric only works with triangle and quad elements.
- `DIAGONAL_RATIO` measures the ratio of the deformed maximum diagonal of an element to the undeformed maximum diagonal of the element. This metric initially has a value of 1.0, and assumes values either greater than or lower than 1 as the mesh deforms. The `DIAGONAL_RATIO` metric only works with hex and quad elements.

The results from any of these distortion metrics can be requested by specifying an `ELEMENT VARIABLES` command line (Section 9.3.1.4) in the `RESULTS OUTPUT` command block (Section 9.3.1) for each metric for which the results are of interest. For example, to request the `ASPECT_RATIO` metric to be output as an element variable named `aspect` in the results output file, the following syntax can be included in a `RESULTS OUTPUT` command block:

```
ELEMENT ASPECT_RATIO as aspect
```

The distortion metrics can also be used for element death (Section 6.5). For example, to kill all elements in which the value of the `SOLID_ANGLE` metric is less than 0.2, the following line can be used in an `ELEMENT DEATH` command block:

```
CRITERION IS ELEMENT VALUE OF SOLID_ANGLE < 0.2
```

For more information about the use of element variables for element death, see Section 6.5.2.2.

2.6 Activation/Deactivation of Functionality

```
ACTIVE PERIODS    = <string list>period_names  
INACTIVE PERIODS = <string list>period_names
```

The `ACTIVE PERIODS` or `INACTIVE PERIODS` command line can be used to activate or deactivate functionality in the code at various points during an analysis. This functionality can include boundary conditions, element blocks, user subroutines, and contact definitions. Command blocks that support this capability are documented accordingly in the sections of this manual where they are described.

In the command line, the string list `period_names` is a list of the time periods defined in `TIME STEPPING BLOCK` command blocks (see Sections 3.1 and 4.10) during which the particular functionality is considered to be active. Each such `period_name` must match a name used in a `TIME STEPPING BLOCK` command block, e.g., `time_block_name`. Each defined time period runs from that period's start time to the next period's start time.

Only one of `ACTIVE PERIODS` or `INACTIVE PERIODS` can be used in any given command block. If the `ACTIVE PERIODS` command line is present, the functionality will be treated as active for all the named periods, and inactive for any time periods that are not listed. If the `INACTIVE PERIODS` command line is present, the functionality will be treated as inactive for all the named periods and active for any period not listed. If neither of these command lines is present, by default the functionality is active during all time periods.

2.7 Error Recovery

```
ERROR TOLERANCE = ON|OFF (ON)
```

This command controls how aggressively Sierra/SM will attempt to recover from a non-physical or ill-defined condition and continue with the analysis.

Under the default setting of `ON`, the code may invoke methods to keep the analysis moving forward despite the presence of non-physical or ambiguous conditions. For example, if the contact algorithm determines that a facet is potentially in two contact surfaces (an ambiguous condition), Sierra/SM will assign the facet to one of the surfaces and continue with the analysis. A second example pertains to SPH particles: if the deformation gradient of an SPH particle is inverted, the particle will be ignored rather than causing the analysis to terminate. Non-physical and ambiguous conditions will generally result in a warning message being printed.

If `ERROR TOLERANCE` is `OFF`, Sierra/SM will terminate an analysis in the presence of ambiguous or non-physical conditions. This may make it easier for an analyst to address non-fatal but potentially problematic issues.

2.8 Manual Job Control

The output of a running job can be controlled externally through the use of “shutdown” and “control” files. This mechanism allows for additional output of restart, results, history, and/or heartbeat data to be requested, as well as an optional graceful shutdown of the job.

A graceful shutdown is requested by inserting a “shutdown” file in the working directory of a running Sierra/SM job. The name of this file can be any of the following:

```
sierra.shutdown  
adagio.shutdown  
<base_name>.shutdown
```

If the `<base_name>.shutdown` variant is used, `<base_name>` is the base name of the input file. For example, if the input file name is `my_analysis.i`, then the shutdown file would be `my_analysis.shutdown`.

If the code detects the existence of a shutdown file, it will dump an output step to any open restart, results, history, or heartbeat file and then gracefully terminate the job. An entry will be written to the log file specifying that a shutdown file was detected and the file will be deleted. The contents of the file are not important; the shutdown file is only checked for existence. Note, the shutdown file will only be looked for at the end of a completed time step. Thus if the solve for a time step is taking an extremely long time the shutdown file will not be detected until that solve completes.

The control file capability is provided to give more control over output and execution than is possible with the shutdown file. The name of the control file is the same as that of the shutdown file except that the file name suffix is `.control` instead of `.shutdown`. The contents of the file consist of a single line, and are case insensitive. The syntax is:

```
DUMP [RESTART] [RESULTS] [HISTORY] [HEARTBEAT] [STEP] [TIME]  
[STOP|ABORT|SHUTDOWN|CONTINUE]
```

The optional strings `RESTART`, `RESULTS`, `HISTORY`, `HEARTBEAT`, `STEP`, and `TIME` are used to specify the type of output that is written before an action is taken. If any output blocks of the specified type were defined in the input file for the model, output will be written to the files. The `STEP` and `TIME` options result in the last step and time being written to the log file. Multiple output types may be requested. If no output type is requested, all types of output specified in the input file will be written.

In addition to controlling the type of output that occurs, the `.control` file also specifies whether the job should be terminated or allowed to continue. If `STOP`, `ABORT`, or `SHUTDOWN` is specified at the end of the line, the job will be gracefully terminated. If `CONTINUE` is specified or no option is specified, the job will continue.

Several examples of the contents of continue files are shown below. The following examples all result in output being written to all types of output file and the job continuing:

```
dump  
dump continue  
dump restart results history heartbeat step continue
```

In both of the following examples, the current step and time will be written to the log file, but no additional output will be written and the job will continue:

```
dump step
dump time continue
```

This example would result in a write to all output types and a graceful shutdown:

```
dump stop
```

In the following example, no output would be written to any files, but the current step and time would be written to the log file before a graceful shutdown:

```
dump step abort
```

An alternate abbreviated syntax is also supported. The abbreviated commands are shown below along with the full commands to which they are mapped:

sw1	dump restart shutdown
sw2	dump step continue
sw3	dump restart continue
sw4	dump results continue

If either the shutdown or control files are used, a message is output to the log file listing the name of that file, and for the control file, the contents of that file. The control or shutdown file is then deleted.

2.9 Solution Termination

Solution termination is the capability to stop the execution when a computed solution reaches some reference solution. This capability can be especially useful for users who want to terminate an analysis when a variable reaches a specific value, but they do not know *a priori* when this will occur. Solution termination may also be used in multi procedure or multi time period analysis to move on to the next solution procedure once some criteria is reached. The inputs are similar to those for solution verification (see Section 9.11).

```
BEGIN SOLUTION TERMINATION
  TERMINATE GLOBAL <string>gVar <|<=|=|>|>=
    <real>gVal [PLUS OR MINUS <real>Tol]
  TERMINATE GLOBAL <string>gVar <|<=|=|>|>=
    GLOBAL <string>gVar2 [PLUS OR MINUS <real>Tol]
  TERMINATE GLOBAL <string>gVar <|<=|=|>|>=
    FUNCTION <string>funcName
  TERMINATE NODAL <string>nVar
    AT NODE <integer>nodeNum <|<=|=|>|>=
    FUNCTION <string>funcName
  TERMINATE ELEMENT <string>eVar
    AT ELEMENT <integer>elemNum <|<=|=|>|>=
    <real>val

  TOLERANCE = <real>absTol
  RELATIVE TOLERANCE = <real>relTol
  TIME TOLERANCE = <real>timeTol

  SKIP TIMES = <real>time0 TO <real>time1
  ACTIVE PERIODS = <string list>periodNames

  COMPLETION FILE = <string>fileName

  TERMINATE TYPE = ENTIRE_RUN|CURRENT_PROCEDURE|
    CURRENT_TIME_PERIOD(ENTIRE_RUN)

  # write results, history, and restart files
  WRITE OUTPUT = OFF|ON(ON)
END
```

The various forms of the TERMINATE GLOBAL, TERMINATE NODAL, and TERMINATE ELEMENT commands check that the named variable has achieved some relation to some defined constant, time history function, or other global variable. Currently, the most thoroughly tested condition is for termination based on a global variable, such as kinetic energy, reaching a constant target value. However, it should be possible to terminate for any variables (including element and nodal), and to allow the target solution to be a user-defined, non-constant variable (e.g., defined via an analytic function that depends on time or other field variables). The global variables used in solution termination can be created via a USER OUTPUT based command block.

Tolerances of how close a solution needs to be to terminate can be specified. The `TOLERANCE` and `PLUS OR MINUS` termination option both specify absolute tolerances. For example the command:

```
TERMINATE GLOBAL disp1 = 1.0 PLUS OR MINUS 0.1
```

will terminate the calculation at the first occurrence of `disp1` being between 0.9 and 1.1.

The `RELATIVE TOLERANCE` command specifies an acceptable relative difference between values for termination. Termination occurs if:

$$\frac{|gVal - gVar|}{\max(|gVal|, |gVar|)} < relTol \quad (2.2)$$

The `TIME TOLERANCE` is used with the time history function based termination commands. A termination will result if the `gVar` is within tolerance to any value of the function that would be computed at the current time plus or minus the `timeTol`.

The `SKIP TIMES` command is used to skip evaluation of whether to terminate over the defined time intervals. This enables solution-based termination to be active only during certain time intervals. Similarly solution termination can be made active only for one or more specified time periods defined in the time stepping block with use of the `ACTIVE PERIODS` command.

The `COMPLETION FILE` command causes solution termination to output a status file named `fileName` at successful completion of the code run. Existence of the status file indicates that the run was terminated successfully. If the termination condition is never reached, the code should run until the final time period.

The `TERMINATE TYPE` command defines what solution termination does when hitting the failure condition. The default `ENTIRE_RUN` option indicates that the entire run should exit when the termination condition is reached. The `CURRENT_PROCEDURE` option indicates that the current procedure will be stopped and the code will move on to the next defined procedure if there is one. If the subsequent procedure defines a start time the next procedure will start at that time. If the subsequent procedure does not define a start time the next procedure will start at the time the current procedure was terminated. The `CURRENT_TIME_PERIOD` option indicates that the current time period (as defined in the time stepping block) will be ended. The Sierra/SM will then move onto the next defined time period if there is one.

The `WRITE OUTPUT` option will write a new results, history, and restart files upon solution termination.



Warning: Multiple results will exist on the output files if output is already scheduled to be written and solution termination also triggers output to be written.

2.9.1 Solution Termination Examples

The following code is used to terminate a run when the global kinetic energy reaches some small value. This may be useful for ending an explicit analysis after a dynamic impact event has occurred and most of the dynamic energy of interest has been dissipated.

```
BEGIN SOLUTION TERMINATION
  TERMINATE GLOBAL kinetic_energy < 1.5e+4
END
```

The following example code could be used end an implicit run immediately before some fracture criteria (eqps) being reached. The fracture itself could then be evaluated in a subsequent explicit procedure.

```
BEGIN USER OUTPUT
  COMPUTE GLOBAL max_eqps AS MAX OF ELEMENT EQPS
  COMPUTE AT EVERY STEP
END

BEGIN SOLUTION TERMINATION
  TERMINATE GLOBAL max_eqps > 0.2
  TERMINATE TYPE = CURRENT_PROCEDURE
END
```



Chapter 3

Explicit Dynamic Time Step Control

This chapter discusses time control for explicit dynamic Sierra/SM analyses. We begin with a broad overview of explicit dynamic time control and then describe the options that are available to users for time control. The commands and features documented in this chapter are applicable only for explicit dynamic analyses. For information on time stepping commands for implicit analyses, see [Chapter 4](#).

The user initiates time control by setting a start time and a termination time for an analysis. The analysis is typically carried out with a large number of time steps, with each time step being much smaller than the analysis time. In explicit transient dynamics analyses, the time step must be less than a critical value. Sierra/SM has a number of methods for computing an estimate for the critical time step. These methods are discussed in detail in this chapter.

The primary time control uses a `TIME CONTROL` command block that appears in the procedure scope. Use of the `TIME CONTROL` command block gives the user, by default, access to an element-based method for estimating the critical time step. The user can access three other methods for estimating the critical time step by using specific command blocks that are placed in the region scope. These other methods tend to give better (larger) estimates for the critical time step. In addition, there is a technique for adjusting the time step known as mass scaling.

Section [3.1](#) describes the `TIME CONTROL` command block. In Section [3.2](#) we discuss the other methods for estimating the critical time step. One approach for improving this estimate is to compute the maximum eigenvalue for a problem. Two methods for computing the maximum eigenvalue are available: the Lanczos method and the power method. Section [3.2.1](#) discusses the Lanczos method; Section [3.2.1.5](#) describes the command block required to implement the Lanczos method. Section [3.2.2](#) discusses the power method; Section [3.2.2.5](#) describes the command block required to implement the power method. Another approach for improving the time step estimate relies on a node-based estimate. Section [3.2.3](#) discusses the node-based method; Section [3.2.3.1](#) describes the command block required to implement the node-based method. You should read the introductory material for the maximum eigenvalue calculation methods and the node-based method and understand this material thoroughly before you attempt to use these methods. Although these other methods give larger time step estimates than the element-based method, they may not result in a net reduction of central processor unit (CPU) time for an analysis unless they are used properly. In those sections dealing with these other methods, we discuss how to use these methods in a cost-effective manner. In Section [3.3](#) we describe the technique of mass scaling, which is a technique that artificially scales up nodal masses to allow larger time steps to be taken, and must be used with the node-based critical time step estimation method.

3.1 Procedure Time Control

As indicated previously, the primary time control uses a `TIME CONTROL` command block in the procedure scope. The user sets the start time and the termination time for the analysis in the `TIME CONTROL` command block. For reference purposes, the general layout of the command block is as follows:

```
BEGIN TIME CONTROL
  BEGIN TIME STEPPING BLOCK <string>time_block_name
    START TIME = <real>start_time_value
    BEGIN PARAMETERS FOR PRESTO REGION <string>region_name
      INITIAL TIME STEP = <real>initial_time_step_value
      TIME STEP SCALE FACTOR =
        <real>time_step_scale_factor(0.9)
      TIME STEP INCREASE FACTOR =
        <real>time_step_increase_factor(1.1)
      STEP INTERVAL = <integer>nsteps(100)
      USER TIME STEP = <real>time_step
      TIME STEP SELECTOR = AUTO
      NUMBER OF QUASISTATIC TIME STEPS =
        <int>quasi_step_count
    END [PARAMETERS FOR PRESTO REGION <string>region_name]
  END [TIME STEPPING BLOCK <string>time_block_name]
  # Second TIME STEPPING BLOCK command block
  # would be placed here, as applicable.
  #
  # Additional TIME STEPPING BLOCK command blocks
  # would be placed here, as applicable.
  #
  # Last TIME STEPPING BLOCK command block
  # would be placed here, as applicable.
  #
  TERMINATION TIME = <real>termination_time
END [TIME CONTROL]
```

The analysis time, as demonstrated above, can be subdivided into a number of time blocks, i.e., `TIME STEPPING BLOCK` command blocks. If the total analysis time is from time 0 to time T and there are three blocks, then the first block is defined from time 0 to time t_1 , the second block is defined from time t_1 to time t_2 , and the third block is defined from time t_2 to time T . (The times t_1 and t_2 are set by the user.) If we sum all the times for each block, the sum will be T . The different time periods defined by each block can be referenced so that we can turn certain functionality on or off throughout an analysis. For example, we may want to have a certain boundary condition turned off during our first time period and activated for the second time period. (Most analyses require only one block.) Use the `ACTIVE PERIODS` or `INACTIVE PERIODS` command lines described in Section 2.6 to activate and deactivate functionality during specific time blocks.

By default, Sierra/SM relies on the element-based critical time step estimate. At every time step, an element-based calculation is performed to determine a critical time step, based on an estimate

of the Courant-Friedrichs-Lewy (CFL) condition that uses the effective material moduli. The effective material moduli can be calculated a number of ways, specified by the `EFFECTIVE MODULI MODEL` command line described in 6.1.6.7. You also have some direct control over the actual time step that is used by employing one of two techniques. We discuss these two techniques in the following paragraphs. The specific command lines for using these techniques are described in Section 3.1.1.

With the `INITIAL TIME STEP` line command you can set an initial time step that is smaller than the element-based critical time step estimate. Sierra/SM will start the analysis by using your initial time step value instead of the element-based critical time step estimate (as long as your value is less than the element-based critical time step estimate). You can then control the rate at which the time step increases from your initial value. This technique is employed by using the `TIME STEP INCREASE FACTOR` command line as follows:

- Setting `TIME STEP INCREASE FACTOR` to 1 causes the specified value of `INITIAL TIME STEP` to be used throughout the analysis (provided that the initial time step is never smaller than the element-based critical time step estimate throughout the computations).
- Setting `TIME STEP INCREASE FACTOR` to a value greater than 1 causes the time step to grow (from the initial time step) at each time step until it reaches the value of the element-based critical time step estimate. From then on, the element-based critical time step estimate will essentially control the time step.

With the second technique, you can manipulate the element-based estimate with either a scale factor or a time step increase factor. This technique is employed by using the `TIME STEP SCALE FACTOR` command line or the `TIME STEP INCREASE FACTOR` command line as follows:

- The element-based estimate for the critical time step is usually smaller than some maximum theoretical value for your model. It may therefore be possible to scale the element-based critical time step estimate by some factor greater than 1. (Your scaled value must remain below the theoretical maximum limit, however. We discuss ways to obtain a critical time step close to the theoretical maximum in later sections of this chapter.)
- If there are stability problems with a particular problem, it may be necessary to scale the element-based estimate with a factor less than 1.
- You can also control the rate at which the time step can increase for an analysis. By specifying a time step increase factor, you can limit the increase in the size of the time step so that it does not increase too rapidly from one step to the next. For certain problems, the element-based critical time step estimate may increase rapidly from one step to the next. Limiting the increase in the size of the time step may enable some problems to run in a more stable fashion.

The `USER TIME STEP` command can be used to explicitly set the integration time step, overriding the critical time step computed by any time step estimation methods. This command can be used to force the analysis to integrate at any specified time step, even a super-critical one.

The `TIME STEP SELECTOR = AUTO` will cause Sierra/SM to select the most advantageous time step from between the standard element based and nodal based time step methods.



Warning: The `TIME STEP SCALE FACTOR` and `USER TIME STEP` commands both provide the ability to force the code to take a time step larger than the stability limit, and thus should be used with caution. If the time step specified with either of these commands is too large, the time integration will add energy to the system, causing the analysis to go unstable and ultimately fail.

Now that we have presented an overview of the functionality in the `TIME CONTROL` command block, we will discuss the actual command lines.

3.1.1 Command Blocks for Time Control and Time Stepping

```
BEGIN TIME CONTROL
  BEGIN TIME STEPPING BLOCK <string>time_block_name
    START TIME = <real>start_time_value
    BEGIN PARAMETERS FOR PRESTO REGION <string>region_name
      #
      # Time control parameters specific to PRESTO
      # are set in this command block.
      #
    END [PARAMETERS FOR PRESTO REGION <string>region_name]
  END [TIME STEPPING BLOCK <string>time_block_name]
  TERMINATION TIME = <real>termination_time
END [TIME CONTROL]
```

Time control resides in a `TIME CONTROL` command block. The command block begins with an input line of the form

```
BEGIN TIME CONTROL
```

and terminates with an input line of the following form:

```
END [TIME CONTROL]
```

Within the `TIME CONTROL` command block, a number of `TIME STEPPING BLOCK` command blocks can be defined. Each `TIME STEPPING BLOCK` command block contains the time at which the time stepping starts and a number of parameters that set time-related values for the analysis. Each `TIME STEPPING BLOCK` command block terminates at the start time of the following command block. The start times for the `TIME STEPPING BLOCK` command blocks must be in increasing order. Otherwise, an error will be generated. (The example in Section 3.1.6 shows the overall structure of the `TIME CONTROL` command block.)

In the above input lines, the values are as follows:

- The string `time_block_name` is a name for the `TIME STEPPING BLOCK` command block. This name must be unique to each command block of this type. The string `time_block_name` can be referenced in an `ACTIVE PERIODS` or `INACTIVE PERIODS` command line to activate and deactivate functionality (see Section 2.6).
- The real value `start_time_value` is the start time for this `TIME STEPPING BLOCK` command block. Values set in this block apply from the start time for this block until the next

start time or the termination time. The start time may be negative.

- The string `region_name` is the name of the Presto region affected by the parameters (see Section 2.2).

The termination time for the analysis is given by the following command line:

```
TERMINATION TIME = <real>termination_time
```

Here, `termination_time` is the time at which the analysis will stop. The `TERMINATION TIME` command line appears inside the `TIME CONTROL` command block but outside of any `TIME STEPPING BLOCK` command block.

The `TERMINATION TIME` command line can appear before the first `TIME STEPPING BLOCK` command block or after the last `TIME STEPPING BLOCK` command block. It is permissible to have `TIME STEPPING BLOCK` command blocks with start times after the termination time; in this case, those command blocks that have start times after the termination time are not executed. Only one `TERMINATION TIME` command line can appear in the `TIME CONTROL` command block. If more than one of these command lines appears, an error is generated.

Nested inside the `TIME STEPPING BLOCK` command block is a `PARAMETERS FOR PRESTO REGION` command block containing parameters that control the time stepping.

```
BEGIN PARAMETERS FOR PRESTO REGION <string>region_name
  INITIAL TIME STEP = <real>initial_time_step_value
  TIME STEP SCALE FACTOR = <real>time_step_scale_factor(0.9)
  TIME STEP INCREASE FACTOR =
    <real>time_step_increase_factor(1.1)
  STEP INTERVAL = <integer>nsteps(100)
END [PARAMETERS FOR PRESTO REGION <string>region_name]
```

These parameters are specific to a Presto analysis.

The command block begins with an input line of the form

```
BEGIN PARAMETERS FOR PRESTO REGION <string>region_name
```

and is terminated with an input line of the following form:

```
END [PARAMETERS FOR PRESTO REGION <string>region_name]
```

As noted previously, the string `region_name` is the name of the Presto region affected by the parameters. The command lines nested inside the `PARAMETERS FOR PRESTO REGION` command block are described next. Some of these command lines will be ignored when either of the maximum-eigenvalue methods or the node-based method is used to estimate the critical time step. The discussions of the command blocks associated with these methods indicate whether these command lines are pertinent.

3.1.2 Initial Time Step

```
INITIAL TIME STEP = <real>initial_time_step_value
```

Presto computes a critical time step for the analysis and uses this value as the initial time step by

default. To directly specify a smaller initial time step, use the `INITIAL TIME STEP` command line, where `initial_time_step_value` is the desired initial time step. This command line is only valid if it is in the first `TIME STEPPING BLOCK` command block in the problem.

The value specified for the initial time step will overwrite the calculated value for the critical time step if the specified initial time step is smaller than the critical time step. If you specify an initial time step that is larger than the critical time step, the time step is set to the value of the calculated critical time step.

3.1.3 Time Step Scale Factor

```
TIME STEP SCALE FACTOR = <real>time_step_scale_factor(0.9)
```

During the element computations, Presto computes a minimum time step required for stability of the computation (the critical time step). Using the `TIME STEP SCALE FACTOR` command line, you can provide a scale factor to modify the critical time step. A value greater than 1.0 for `time_step_scale_factor` will cause the time step to be greater than the computed critical time step, and thus the problem may become unstable. By default, the scale factor is 0.9.

3.1.4 Time Step Increase Factor

```
TIME STEP INCREASE FACTOR =  
  <real>time_step_increase_factor(1.1)
```

During an analysis, the computed critical time step may change as elements deform, are killed, and so forth. By using the `TIME STEP INCREASE FACTOR` command line, you can limit the amount that the time step can increase between two adjacent time steps. The value `time_step_increase_factor` is a factor that multiplies the previous time step. The current time step can be no larger than the product of the previous time step and the scale factor.

An increase factor less than 1.0 will cause the time step to continuously decrease. The default value for this factor is 1.1, i.e., a time step cannot be more than 1.1 times the previous step.

3.1.5 Step Interval

```
STEP INTERVAL = <integer>nsteps(100)
```

Presto can output data about the current time step, the current internal and external energy, and the kinetic energy throughout an analysis to the log file. The `STEP INTERVAL` command line controls the frequency of this output, where `nsteps` is the number of time steps between output. The default value for `nsteps` is 100.

The output at any given step (read from left to right) is

- step number,
- time,
- time step,
- global element identifier for element controlling time step

- kinetic energy,
- internal energy,
- external energy,
- error in energy balance,
- hourglass energy
- CPU time, and
- wall clock time.

The time is at the current time, step n , and the time increment is the previous time step increment from step $n - 1$ to step n .

The error in the energy balance is computed from the following relation:

$$\text{energy balance error} = (\text{kinetic energy} + \text{internal energy} - \text{external energy}) / \text{external energy} * 100$$

The above expression gives a percent error for the energy balance.

3.1.6 Time Control Example

The following is a simple example of a `TIME CONTROL` command block:

```
BEGIN TIME CONTROL
  BEGIN TIME STEPPING BLOCK p1
    START TIME = 0.0
    BEGIN PARAMETERS FOR PRESTO REGION presto_region
      INITIAL TIME STEP = 1.0e-6
      STEP INTERVAL = 50
    END
  END
  BEGIN TIME STEPPING BLOCK p2
    START TIME = 0.5e-3
    BEGIN PARAMETERS FOR PRESTO REGION presto_region
      TIME STEP SCALE FACTOR = 0.9
      TIME STEP INCREASE FACTOR = 1.5
      STEP INTERVAL = 10
    END
  END
  TERMINATION TIME = 1.0e-3
END
```

The first `TIME STEPPING BLOCK`, `p1`, begins at time 0.0, the initial start time, and terminates at time 0.5×10^{-3} . The second `TIME STEPPING BLOCK`, `p2`, begins at time 0.5×10^{-3} and terminates at time 1.0×10^{-3} , which is the time listed on the `TERMINATION TIME` command line. The `TIME STEPPING BLOCK` names `p1` and `p2` can be referenced by the `ACTIVE PERIODS` or `INACTIVE PERIODS` command lines as described in [Section 2.6](#) to activate and deactivate functionality.

3.1.7 Explicit Quasistatic Mode

```
BEGIN PARAMETERS FOR PRESTO REGION <string>region_name
  NUMBER OF QUASISTATIC TIME STEPS = <int>quasi_step_count
END [PARAMETERS FOR PRESTO REGION <string>region_name]
```

Explicit quasistatic mode provides an automated methodology to leverage explicit time stepping to efficiently solve quasistatic problems. Quasistatic mode is enabled by providing a non-zero input for the `NUMBER OF QUASISTATIC TIME STEPS` command for a solution time period. Explicit solution of a quasistatic analysis may prove to be more robust or faster than implicit solution in some cases.

Explicit quasistatic mode can be particularly helpful for analyses with contact. Analyses with several disjoint parts that are in static equilibrium only due to the contact and frictional forces between those may cause great difficulty for the standard implicit static solver. Explicit quasistatic mode can often solve these problems more robustly than the implicit solver. Additionally explicit quasistatic mode is well suited to analyses that involve buckling, strong material nonlinearity, and material failure or fracture. Such sharp nonlinearity cause substantial difficulty for standard implicit solution.

Activating explicit quasistatic mode has several effects on the analysis.

- Implicit compatible defaults are used for element formulations for numerical parameters such as effective modulus, quadratic bulk viscosity, and linear bulk viscosity.
- The analysis is automatically mass scaled (See Section 3.3) so that `quasi_step_count` steps will be taken in the current time period. As dynamic terms still have some meaning in explicit quasistatic analysis to function efficiently the size of the implied quasistatic time step should be significantly larger than the standard explicit time step. For example if the standard explicit time step is $1.0\text{e-}6$ seconds and it is desired the load up quasistatically over 100,000 steps then the time over which to perform this loading should be one second or greater. This would yield a quasistatic time step of $1.0\text{e-}5$ seconds which is significantly larger than the standard explicit time step.
- To minimize dynamic effects explicit quasistatic analysis should be run at a time step that exceeds the explicit time step. Sometimes it is convenient to greatly exceed the standard explicit time step. For stability reasons the time step is slowly increased from the standard explicit time step when entering a quasistatic solution period and slowly decreased to the standard explicit time step when exiting a quasistatic solution period.
- A dynamic viscous damping term is included to accelerate convergence to a static equilibrium solution. This viscous term is automatically tuned based on the number of time steps being taken and the current model velocity.
- A static residual norm is automatically computed and output to the global variable `quasistatic_residual`. Monitoring of this residual can indicate how closely the current solution approximates the target quasistatic state. The quasistatic residual is the L2 norm of the current nodal force imbalance divided by the the reference force (reaction or external force sum). The explicit quasistatic residual has the same meaning as the implicit relative residual, generally a good solution will have a quasistatic residual of 10^{-3} or below.

3.1.7.1 Usage Guidelines

It is important to note that quasistatic mode is a *dynamic* analysis; dynamic terms and effects will be present in the solution. However, if used properly, quasistatic mode will minimize these effects, yielding a solution with comparable accuracy to an equivalent implicit static analysis. Generally, the number of quasistatic explicit steps required to reach static equilibrium will be tens to hundreds of thousands, although a sharp estimate of the actual number of steps required is not available. Generally, dynamic effects are expected to reduce with increasing numbers of time steps. The quasistatic residual norm variable may be used to monitor static equilibrium at various steps of the analysis.

Quasistatic equilibrium is reached fastest on models with minimal potential low-mode vibrations. The most difficult types of analyses for quasistatic mode to solve include long, slender structural members that can vibrate at low frequencies.

Static equilibrium in explicit quasistatic mode is also sensitive to *loading rate*; more smoothly applied external loads will result in more rapid convergence to the quasistatic solution. It is also recommended that loads be held constant in the later part of the loading period. Ultimate quasistatic equilibrium can be reached in the contact load. An example showing recommended quasistatic mode loading is shown in Section [3.1.7.2](#).

Output results from quasistatic mode—particularly quantities such as reaction force and energy—are valid only when the model has reached static equilibrium. The mass scaling used by quasistatic mode can alter the results in the middle of a load step. Damping and other artificial forces may be arbitrarily large when the model is in motion in an intermediate state, while they tend to zero when a model reaches static equilibrium.

It is assumed that the ultimate material state obtained is strain path-independent as long as the strain path is monotonic. This condition holds for elastic models, J_2 plasticity models, soil foam, and most other commonly used models, but does *not* hold for rate-dependent models. Quasistatic mode will provide an answer for rate-dependent models, but a very large number of load steps may be required to ensure the loading is applied smoothly.

Although the smoothness of the strain paths and reduction of dynamic effects will improve with iterations, the most reliable way to detect possible material state overshoot error is to compare the result with a solution based on a larger number of quasistatic iterations. Current evidence indicates quasistatic material state integration errors reduce quadratically in iteration count.

3.1.7.2 Example Problem

Consider following example problem demonstrates the use of explicit quasistatic mode for a beam loaded by a pressure boundary condition. Some important part of the input deck are shown bellow.

```
begin function pfunc
  evaluate expression = cos_ramp(t, 0.0, 0.5)
end

begin time control
  begin time stepping block pl
```

```
        start time = 0.0
        begin parameters for presto region presto
            number of quasistatic time steps = 2000
        end
    end
    termination time = 1.0
end

begin presto region
    begin pressure
        surface = surface_1000
        function = pfunc
    end
end
```


3.2 Other Critical Time Step Methods

Currently, there are four methods for calculating a critical time step for Presto. First, there is the traditional element-based method. We know that, in general, the element-based time step in Presto can give a fairly conservative estimate for the time step. Second, there is a node-based method for giving a critical time step estimate. Depending on the problem, the node-based method may or may not give an estimate for the critical time step that approaches the theoretical maximum value for a particular model. Although the node-based method can give a larger critical time step estimate than the element-based method, the node-based estimate may still be significantly lower than the maximum theoretical time step for a problem. Finally, there are two methods that use an estimate for the maximum eigenvalue to obtain an estimate for the critical time step. From the maximum eigenvalue, it is possible to derive the theoretical maximum critical time step for a problem via the formula

$$\Delta t_{crit} = 2.0 / \sqrt{\theta_{max}}, \quad (3.1)$$

where Δt_{crit} is the critical time step and θ_{max} is the maximum eigenvalue. The two methods employed in Presto to give an estimate for the maximum eigenvalue are the Lanczos method and the power method. The Lanczos method can give an accurate estimate of the maximum eigenvalue for a problem using a very small number of Lanczos vectors compared to the total number of degrees of freedom in a problem. The power method uses a simple iterative process to obtain an estimate for the maximum eigenvalue. The power method is not as powerful a mathematical technique as the Lanczos method for obtaining a maximum eigenvalue estimate. However, it does present another viable option for obtaining a maximum eigenvalue estimate for certain problems and has therefore been included as another method for obtaining the maximum eigenvalue estimate. See [1, 2] for more information on the Lanczos method and other related methods in computational linear algebra.

The use of a critical time step from the node-based method or a method based on the maximum eigenvalue estimate is desirable because the larger critical time steps produced by these methods (compared to the element-based method) reduce CPU time. Both methods, however, are not cost effective if they are called every time step to give a critical time step estimate. The cost of doing one node-based estimate or one maximum eigenvalue estimate for the critical time step will not offset the cost benefit of the increase to the critical time step (compared to the element-based time step estimate) over a single time step. Hence, there must be some scheme for

- calling these methods only periodically throughout a calculation and
- maintaining a larger estimate (than the element-based estimate) for the critical time step in between these calls

if we are to gain a net benefit from the increase in the critical time step these methods can produce.

If you want to use the maximum eigenvalue estimate for calculating the critical time step and your problem has long periods with a relatively stable time step estimate, the Lanczos method for calculating the maximum eigenvalue will be the preferred method to use. If you have a problem for which the changes in the time step should be monitored more frequently, the power method will be better suited for this problem than the Lanczos method. The preferred use for each of these two different eigenvalue methods will become obvious as you read the background material for these two methods in Section 3.2.1 and Section 3.2.2. Both the Lanczos method and the power

method require the use of a variety of control parameters. For some of these control parameters, techniques have been devised to automatically calculate values for these parameters in what should be a robust and reliable method for a wide range of problems. Other parameters are defaulted to values that will optimize the use of a particular eigenvalue calculation method (Lanczos method or power method) with the type of problems that best suit the method that is employed. The default values for these control parameters may change as we gain more experience in using the Lanczos method and the power method.

Detailed discussions of the Lanczos method, the power method, and the node-based method follow. The implementation of the Lanczos method and the power method have many similarities.

3.2.1 Lanczos Method

The Lanczos method, as implemented in Presto, is outlined here so that certain aspects of it can be referenced in subsequent parts of this chapter. In the following set of equations, $\mathbf{K}_T$ is a tangent stiffness matrix, $\mathbf{M}$ is the mass matrix, and $\mathbf{r}_0$ is an arbitrary starting vector.

Initialize

$$\mathbf{q}_0 = \mathbf{0}$$

$$\beta = ([\mathbf{r}_0]^T \mathbf{M}^{-1} \mathbf{r}_0)^{1/2}$$

$$\mathbf{q}_1 = \mathbf{r}_0 / \beta_1$$

$$\mathbf{p}_1 = \mathbf{M}^{-1} \mathbf{q}_1$$

for $j = 1, n$

$$\bar{\mathbf{r}}_j = \mathbf{K}_T \mathbf{p}_j$$

$$\hat{\mathbf{r}}_j = \bar{\mathbf{r}}_j - \mathbf{q}_{j-1} \beta_j$$

$$\alpha_j = [\mathbf{q}_j]^T \mathbf{M}^{-1} \hat{\mathbf{r}}_j = [\mathbf{p}_j]^T \hat{\mathbf{r}}_j$$

$$\mathbf{r}_j = \hat{\mathbf{r}}_j - \alpha_j \mathbf{q}_j$$

$$\hat{\mathbf{p}}_j = \mathbf{M}^{-1} \mathbf{r}_j$$

$$\beta_{j+1} = ([\mathbf{r}_j]^T \mathbf{M}^{-1} \mathbf{r}_j)^{1/2} = ([\hat{\mathbf{p}}_j]^T \mathbf{r}_j)^{1/2}$$

if enough vectors, terminate loop

$$\mathbf{q}_{j+1} = \mathbf{r}_j / \beta_{j+1}$$

$$\mathbf{p}_{j+1} = \hat{\mathbf{p}}_j / \beta_{j+1}$$

end

The details for this form of the Lanczos method are described in Reference [3]. Notice that the Lanczos method is an iterative method. If we use the Lanczos method in a code like Presto to compute the maximum eigenvalue for a particular finite element model, the number of iterations required to give a good estimate for the maximum eigenvalue will depend on the size of the finite element model (the number of nodes and elements), the types of elements in the model, and the material types and material properties used in the model.

The maximum eigenvalue for a particular finite element model gives us the largest estimate we can obtain for the critical time step. Hence, the time step estimate derived from the maximum eigenvalue is our “best” estimate for the critical time step. The estimate for the critical time step based on the maximum eigenvalue can be significantly larger than an element-based time step estimate. However, computing the critical time step (for a given time step) with the Lanczos method is more expensive than computing the critical time step with element-based calculations. Over one time step, it is not possible to recoup the cost of the Lanczos calculations with the increase in the size of the time step over the element-based estimate. Using the Lanczos method for estimating the critical time step in an explicit, transient dynamics code requires a methodology that effectively addresses the computational costs. The following sections outline a cost-effective approach to using the Lanczos method in an explicit, transient dynamics code.

3.2.1.1 Lanczos Method with Constant Time Steps

To explain how to use the Lanczos method in a cost-efficient manner, we must first establish the computational cost of using the Lanczos method. As indicated previously, computing the maximum eigenvalue for a finite element model requires some number of iterations (each iteration in the Lanczos method produces a Lanczos vector) to obtain a good estimate for the maximum eigenvalue. The cost of an iteration (Lanczos vector) is approximately the cost of an internal force calculation. Notice, in preceding equations for the Lanczos method, that the Lanczos method requires the product of the tangent stiffness matrix $\mathbf{K}_T$ with a vector $\mathbf{p}$. In Presto, we do not construct a tangent stiffness matrix. Instead, we simply provide the vector $\mathbf{p}$ for the internal force calculations. The internal force calculations give us the desired matrix $\times$ vector product of $\mathbf{K}_T\mathbf{p}$.

Over a given time step, the cost of an internal force calculation is the major computational cost. (This assumes no contact. The addition of contact introduces another computationally expensive process into a time step. For our initial discussion, we ignore the cost of contact.) To use the Lanczos method to get a critical time step estimate, one has to call the Lanczos method for some given time step. If the Lanczos method made n iterations to get a good estimate for the maximum eigenvalue, then the overall cost of the time step would be approximately $n + 1$ times the cost of the internal force calculation. The cost of the time step would be the n internal force calculations for Lanczos and the actual internal force calculation to advance the time step. If n is 20 (a minimum for typical problems), the cost of the time step becomes 21 times the cost of the internal force calculation. The critical time step estimate based on the maximum eigenvalue would have to be at least 21 times greater than the element-based critical time step to recoup the cost of the maximum eigenvalue calculation. A typical value for the critical time step estimate based on the maximum value is more in the range of 1.1 to 2.0 times the element-based critical time step estimate. Obviously, the Lanczos method is too expensive to call at every time step for a critical time step estimate.

To explain how the Lanczos method can effectively be used in an explicit, transient dynamics code, we begin with a simple case study. In this case study, we compute the critical time step using the Lanczos method at some time step and then assume that this critical time step value remains constant for a subsequent number of time steps, n_L . We only call the Lanczos method once during the n_L time steps. (In reality, the critical time step in an explicit, transient dynamics code like Presto changes with each time step. We address this issue of the changing time step when we present the

details for using the Lanczos method in a cost-effective manner.) For this case study, we also assume that the computational cost of an element-based estimate for the critical time step is part of the cost of an internal force calculation. The cost of the element-based time step estimate is a small part of the overall internal force calculations. Finally, for our initial discussion, we assume no contact. We address the issue of contact further in the discussion.

Assume that the Lanczos method computes a global estimate for the critical time step of Δt_L , which is the value to be used for n_L time steps. At the end of the n_L time steps, the analysis time for the code has been incremented by an amount ΔT , which is computed simply as

$$\Delta T = n_L \Delta t_L. \quad (3.2)$$

If the element-based estimate for the time step is Δt_e and the number of time steps required to increment the analysis time by ΔT is n_e , then, for the element-based time step, we have

$$\Delta T = n_e \Delta t_e. \quad (3.3)$$

Because the Lanczos estimate for the critical time step is larger than the element-based estimate, we know that $n_e > n_L$. Let us define the ratio r as

$$r = \Delta t_L / \Delta t_e = n_e / n_L. \quad (3.4)$$

The ratio r is greater than 1. Now that we have determined the relation between the number of steps required for a Lanczos-based critical time step estimate versus the element-based critical time step estimate to achieve the same analysis time increment, let us examine the computational costs for these two cases in terms of CPU time. Designate the CPU cost for a time step as Δt_{IF} . If the number of Lanczos vectors required to obtain the critical time step estimate is N_L , then the total computational cost of the Lanczos method and the n_L time steps is

$$n_L \Delta t_{IF} + N_L \Delta t_{IF}. \quad (3.5)$$

If we use the element-based method, the total computational cost is

$$n_e \Delta t_{IF}. \quad (3.6)$$

Recall that we have chosen n_L and n_e so that we have the same analysis time increment ΔT even though we have different critical time steps. Now, we must determine the point at which the computational cost for the Lanczos-based critical time step calculations is the same as the cost for the element-based critical time step calculations. This is simply the point at which

$$n_e \Delta t_{IF} = n_L \Delta t_{IF} + N_L \Delta t_{IF}. \quad (3.7)$$

If we rearrange the above equation to eliminate Δt_{IF} and make use of the ratio r , then we obtain

$$n_e = \frac{N_L}{1 - 1/r}. \quad (3.8)$$

Consider the case of $r = 1.25$ and $N_L = 20$. When $n_L = 80$ and $n_e = 100$, the above equations show that the calculations with the Lanczos-based critical time step and the calculations with the

element-based time step give the same analysis time for the same computational expense. If we use the Lanczos-based critical time step Δt_L for more than eighty iterations, then the Lanczos-based approach becomes cost effective. Our above equations have established the "break-even" point at which it becomes cost effective to use the Lanczos method to reduce computational costs by overcoming the initial cost of the Lanczos calculations with the larger critical time step.

We can build on what we have done thus far to account for contact. Suppose that the computational cost of contact over a time step is some multiple m of the computational cost of the internal force calculation Δt_{IF} . Then the point at which the computational cost for the Lanczos-based calculations is the same as the computational cost for the element-based calculations is

$$(1 + m)n_L\Delta t_{IF} + N_L\Delta t_{IF} = (1 + m)n_e\Delta t_{IF}. \quad (3.9)$$

For the case with contact,

$$n_e = \frac{N_L}{(1 + m)(1 - 1/r)}. \quad (3.10)$$

Again, consider the case of $r = 1.25$ and $N_L = 20$. Assume the computational cost of contact calculations is the same as an internal force calculation ($m = 1$). For these values, the break-even point is $n_L = 40$ and $n_e = 50$. The added computational cost of the contact calculations results in reaching the break-even point with a smaller number of iterations when compared to the case with no contact.

The above derivations let us calculate a break-even point based on our assumptions of a constant critical time step. Considering that a typical analysis will run for tens of thousands of time steps, something on the order of 100 steps represents a reasonable number of steps to recoup the cost of the Lanczos calculations. Whether the cost of the Lanczos calculations can be recouped in something on the order of 100 calculations depends heavily upon N_L . If N_L is sufficiently small, we can recoup the cost of the Lanczos calculations in a reasonable number of steps.

Some computational studies indicate that N_L is in an acceptable range for many problems. The Lanczos method computes a good estimate for the maximum eigenvalue with a small number of Lanczos vectors, N_L , compared to the number of degrees of freedom in a problem. Some component studies show that, for a problem with between 250,000 and 350,000 degrees of freedom, one can obtain a good estimate for the maximum eigenvalue with only twenty Lanczos vectors. A large-scale study of a model involving 1.7 million nodes (5.1 million degrees of freedom) showed that only forty-five Lanczos vectors were required to obtain a good estimate of the maximum eigenvalue. These examples demonstrate that the number of Lanczos vectors required for a good maximum eigenvalue estimate is very small when compared to the number of degrees of freedom for a problem. When N_L is in the range of twenty to forty-five, Equations (3.7) and (3.10) show that, with an increase in the time step of about 1.2 to 1.25, we can recoup the cost of the Lanczos method in a reasonable number of time steps.

Now that we have determined we can recoup the cost of the Lanczos calculations in a reasonable number of time steps, let us look at the issue of reusing a Lanczos-based estimate in some manner.

3.2.1.2 Controls for Lanczos Method

As indicated in the above discussion, the Lanczos method can be used in a cost-effective manner in an explicit, transient dynamics code if a Lanczos calculation can be performed and the Lanczos-

based estimate for the critical time step can be reused in some way over many subsequent time steps. This section presents an approach for reusing a Lanczos-based estimate over a number of time steps so that we maintain a critical time step estimate that is close to the theoretical maximum value in between the calls to the Lanczos method. The approach discussed here makes use of the element-based critical time step estimate at each time step.

We start our approach with a Lanczos calculation to determine the maximum eigenvalue. The Lanczos method converges to the maximum eigenvalue from below, which means that the method underestimates the maximum eigenvalue. Because the critical time step depends on the inverse of the maximum eigenvalue, we overestimate the critical time step. It is necessary, therefore, to scale back the critical time step estimate from the Lanczos method so that the calculations in the explicit time-stepping scheme do not become unstable. Our approach for determining a scaled-back value for the maximum critical time step makes use of the element-based time step estimate. Again, let Δt_L be the critical time step estimate from the Lanczos method and Δt_e be the critical time step estimate from the element-based calculations. The scaled-back estimate for the critical time step, Δt_s , is computed from the equation

$$\Delta t_s = \Delta t_e + f_s(\Delta t_L - \Delta t_e), \quad (3.11)$$

where f_s is a scale factor. (A reasonable value for f_s ranges from 0.9 to 0.95 for our problems.) This value of f_s puts Δt_s close to and slightly less than a theoretical maximum critical time step.

Once Δt_s is determined, the ratio

$$t_r = \Delta t_s / \Delta t_e \quad (3.12)$$

is computed. This ratio is then used to scale subsequent element-based estimates for the critical time step. If $\Delta t_{e(n)}$ is the n^{th} element-based critical time step after the time step where the Lanczos calculations are performed, then the n^{th} time step after the Lanczos calculations, $\Delta t_{(n)}$, is simply

$$\Delta t_{(n)} = t_r \Delta t_{e(n)}. \quad (3.13)$$

The ratio t_r is used until the next call to the Lanczos method. The next call to the Lanczos method is controlled by one of two mechanisms. With the first mechanism, the user can set the frequency with which the Lanczos method is called. The user can set a parameter so that the Lanczos method is called only once every n time steps. This number remains fixed throughout an analysis. With the second mechanism, the user can control when the Lanczos method is called based on changes in the element-based critical time step. For this second mechanism, the change in the element-based critical time step estimate is tracked. Suppose the element-based critical time step at the time the Lanczos method was called is Δt_e . At the n^{th} step after the call to the Lanczos method, the element-based critical time step is $\Delta t_{e(n)}$. If the value

$$\Delta t_{lim} = \frac{|\Delta t_{e(n)} - \Delta t_e|}{\Delta t_e} \quad (3.14)$$

is greater than some limit set by the user, then the Lanczos method will be called. If there is a small, monotonic change in the element-based critical time step over a large number of time steps, this second mechanism will result in the Lanczos method being called. Or if there is a large, monotonic change in the element-based critical time step over a few time steps, the Lanczos method will also be called.

These two mechanisms for calling the Lanczos method can be used together. For example, suppose the second mechanism (the mechanism based on a change in the element-based time step) results in a call to the Lanczos method. This resets the counter for the first mechanism (the mechanism using a set number of time steps between calls to the Lanczos method).

This approach for reusing a Lanczos-based time step estimate has been implemented in Presto, and used for a number of studies. One of the component studies, as indicated previously, used the same scale factor for $n_L = 1700$ iterations. The break-even point for this problem is $n_e = 45$ time steps (not accounting for contact, which was a part of the component modeling). For this particular problem, the extended use of the Lanczos estimate reduced the computational cost to 56% of what it would have been with the element-based time step.

Not all problems will lend themselves to reusing one Lanczos-based estimate for thousands of time steps. However, if it is possible to use the Lanczos-based estimate for two to three times the number of time steps required to reach the break-even point, we begin to see a noticeable reduction in the total number of time steps required for a problem.

3.2.1.3 Scale Factor for Lanczos Method

When the Lanczos method is called for a given time step, it must appear that the calculations are using the constant tangent stiffness matrix $\mathbf{K}_T$ for all iterations. As indicated previously, we use the internal force calculations to generate the product $\mathbf{K}_T \mathbf{p}_j$ (for the j^{th} iteration) in the Lanczos calculations. Any vector $\mathbf{p}_j$, as calculated by the Lanczos method, may be such that it represents large-strain behavior and moves the internal force calculations into a nonlinear regime. It is necessary to scale the $\mathbf{p}_j$ vectors so that the internal force calculations are in a small-strain regime, which makes it appear that we are working with a constant tangent stiffness matrix. The vectors $\mathbf{p}_j$ must be scaled so that they represent velocities associated with small strain. When properly scaled vectors are sent to the internal force calculation, the internal force calculation effectively becomes a matrix $\times$ vector product with a constant tangent stiffness matrix for all iterations during a given call to the Lanczos method.

The scale factor for the $\mathbf{p}_j$ vectors, which we will designate as v_{sf} , must not be too small, as this will create round-off problems and give a bad estimate for the critical time step. If the scale factor is too large, we violate the above restriction of a constant tangent stiffness matrix.

There are two approaches for controlling the scale factor when the Lanczos method is used to compute the maximum eigenvalue. These approaches are discussed in Section 3.2.1.5.

3.2.1.4 Accuracy of Eigenvalue Estimate

Every time a new Lanczos vector is computed, we obtain an additional eigenvalue for our model and, in general, a better estimate for the maximum eigenvalue (and hence a better estimate for the critical time step). The Lanczos method can compute a good value for the maximum eigenvalue with a very small number of total computed eigenvalues compared to the number of degrees of freedom in a problem. There are examples, as previously indicated, of problems with 250,000 to 350,000 degrees of freedom where we have obtained a good estimate of the critical time step with twenty eigenvalues. In one problem with 5.1 million degrees of freedom, we obtained a good estimate of the critical time step with forty-five eigenvalues.

A user could, in theory, determine a reasonable number of eigenvalues necessary for obtaining a good estimate of the maximum eigenvalue based on the above information on model size and the number of eigenvalues required for a good maximum eigenvalue estimate. The user could test the validity of the choice of the number of eigenvalues by increasing the number of eigenvalues slightly and comparing the maximum eigenvalue estimate obtained with the larger number of eigenvalues to the original maximum eigenvalue estimate obtained with the smaller number of eigenvalues. If the change in the two maximum eigenvalue estimates (larger versus smaller total number of eigenvalues) is small, then the original estimate for the number of eigenvalues is reasonably accurate.

As an alternative to directly specifying the number of eigenvalues to be computed, a convergence tolerance could be set on the change in the magnitude of the maximum eigenvalue as additional eigenvalues (Lanczos vectors) are computed. Let θ_{max_n} be the maximum eigenvalue calculated corresponding to n eigenvalues (Lanczos vectors), and let θ_{max_n+1} be the maximum eigenvalue corresponding to $n + 1$ eigenvalues (Lanczos vectors). The eigenvalues would be computed until

$$\frac{|\theta_{max\_n+1} - \theta_{max\_n}|}{\theta_{max\_n+1}} \quad (3.15)$$

is less than some tolerance. (We will now refer to the value of Equation (3.15) as the *convergence measure*.) If we calculate the convergence measure for a sequence of maximum eigenvalues computed by the Lanczos method, we will not necessarily see a monotonic decrease in the convergence measure for all time. Typically, the convergence measure will initially show a monotonically decreasing value for an increasing n , and then the convergence measure will show a monotonically increasing value for an increasing n . (The phase in which the convergence measure increases is usually very small, about one to two iterations.) After the phase in which the convergence measure monotonically increases, there is then a phase in which the convergence measure begins a long (over many iterations) monotonic decrease. This behavior (decrease of the convergence measure, slight increase, then long monotonic decrease) is typical of Krylov methods, of which the Lanczos method is one. Using the convergence measure to determine the number of eigenvalues to be computed for a problem is a reasonable option, but some care must be taken in setting the tolerance for the convergence measure.

Both of the techniques discussed above are offered as a way to set the maximum number of eigenvalues required to obtain an accurate eigenvalue estimate for a model. These techniques are discussed further in Section 3.2.1.5.

3.2.1.5 Lanczos Parameters Command Block

```
BEGIN LANCZOS PARAMETERS <string>lanczos_name
  STARTING VECTOR = <string>STRETCH_X|STRETCH_Y|STRETCH_Z|
    ISOTHERMAL (ISOTHERMAL)
  INCREASE OVER STEPS = <integer>incr_int(5)
  NUMBER EIGENVALUES = <integer>num_eig(100)
  EIGENVALUE CONVERGENCE TOLERANCE = <real>converge_tol(0.5e-3)
  SMALL STRAIN = <real>small_strain(1.0e-3)
  VECTOR SCALE = <real>vec_scale(1.0e-5)
  SCALE FACTOR = <real>time_scale(1.0)
  UPDATE ON TIME STEP CHANGE = <real>tstep_change(0.10)
```



```

UPDATE STEP INTERVAL = <integer>step_int(500)
FORCE LANCZOS TIMESTEP = <string>ON|OFF(OFF)
END [LANCZOS PARAMETERS <string>lanczos_name]

```

If you use the Lanczos method to compute a critical time step, there should be only one LANCZOS PARAMETERS command block, and it should appear in the region. If you have a LANCZOS PARAMETERS command block, you should not specify a NODE BASED TIME STEP PARAMETERS or a POWER METHODS PARAMETERS command block. If you use the Lanczos method to compute the critical time step, the time step increase factor (default or user-specified) will be used to control the increase in the time step estimate; the time step scale factor (default or user-specified) will be used to scale the final time step. These factors are specified in the PARAMETERS FOR PRESTO REGION portion of a TIME STEPPING BLOCK in the TIME CONTROL command block.

The Lanczos method requires some type of starting vector. This is determined by the STARTING VECTOR command line. The various options available for this command line will generate a displacement vector that stretches your model in the x -, y -, or z -direction, or in all three directions (x , y , and z) at once. (STRETCH_X stretches the model in the x -direction, STRETCH_Y stretches the model in the y -direction, STRETCH_Z stretches the model in the z -direction, and ISOTHERMAL stretches the model in all three directions at once.) The displacement vector then serves as a basis for generating the starting vector r_0 in the initialization phase of the Lanczos method. The Lanczos method appears to be fairly insensitive to the choice of a starting vector. The default starting vector is the ISOTHERMAL option. You may encounter cases where the use of one of the other options for a starting vector—STRETCH_X, STRETCH_Y, or STRETCH_Z—may result in a slight increase in accuracy for the critical time step estimate for a given number of eigenvalues. These other options (STRETCH_X, STRETCH_Y, and STRETCH_Z) are offered for these special cases.

The INCREASE OVER STEPS command line determines how many steps, i.e., `incr_int`, are used to transition from an element-based critical time step estimate to the Lanczos-based estimate at the beginning of an analysis. The user may want to increase from the element-based estimate to the Lanczos-based estimate over a number of time steps if the difference between these two estimates is large. The value of `incr_int` for this command line defaults to 5.

As indicated in Section 3.2.1.4, some number of eigenvalues must be computed by the Lanczos method to obtain a good estimate for the maximum eigenvalue for a given finite element model. As one option, the user can simply specify the number of eigenvalues to be computed by the Lanczos method by using the NUMBER EIGENVALUES command line. The default value for `num_eig` in this command line is 100. As an alternative, the user can specify a tolerance on the convergence measure defined in Equation (3.15). The tolerance is set with the EIGENVALUE CONVERGENCE TOLERANCE command line. The default value for the convergence tolerance, `converge_tol`, is 0.5×10^{-3} . (This default value may change in the future as we gain more experience with the Lanczos method.)

By default, the convergence measure (with the default value of 0.5×10^{-3}) is used to determine the number of eigenvalues to be computed. If neither the EIGENVALUE CONVERGENCE TOLERANCE command line nor the NUMBER EIGENVALUES command line appears in the LANCZOS PARAMETERS command block, then the convergence measure will be used to deter-

mine the number of eigenvalues. If the `EIGENVALUE CONVERGENCE TOLERANCE` command line appears in the command block with a specified tolerance other than the default value, then the number of eigenvalues computed will be determined by the convergence measure and the user-specified tolerance. If the `NUMBER EIGENVALUES` command line appears in the `LANCZOS PARAMETERS` command block, then the user-defined number of eigenvalues will be used to set the number of eigenvalues computed by the Lanczos method.

The recommended technique for determining the number of eigenvalues is to use the convergence measure option with the default tolerance. If you use this technique, neither the `NUMBER EIGENVALUES` command line nor the `EIGENVALUE CONVERGENCE TOLERANCE` command line should be included in the `LANCZOS PARAMETERS` command block.

As indicated in Section 3.2.1.3, it is necessary to scale one of the vectors (with a scale factor v_{sf}) in the Lanczos calculations so that it appears we are always working with a constant tangent stiffness each time we call the Lanczos method. There are two approaches for determining the scale factor. With the first approach, the user can specify the value of a small strain for a particular model by using the `SMALL STRAIN` command line. (As long as the internal force calculation inside the Lanczos method is working with a vector that approximates small-strain behavior, it will appear that we are working with a constant tangent stiffness inside the Lanczos method.) The value for v_{sf} is then computed based on your specification for what constitutes a small strain in the model. (The default value for `small_strain` is 1.0×10^{-3} .) With the second approach for determining the scale factor, the user can specify a value for v_{sf} directly by using the `VECTOR SCALE` command line. The default value for v_{sf} , the `vec_scale` parameter in that command line, is 1.0×10^{-5} . Many tests have established that this value for the scale factor, 1.0×10^{-5} , works well for a range of models encountered at Sandia National Laboratories. If the user wants to use the option of directly specifying the scale factor, it is possible to determine whether a particular scale factor is suitable for a particular problem. Take a scale factor v_{sf} , plus values on either side of it, say, $0.9 \times v_{sf}$ and $1.1 \times v_{sf}$. If all three of these scale factor values produce almost the same estimate for the critical time step for a particular model, then the value for v_{sf} meets the criterion for an acceptable scale factor.

By default, the small-strain approach (with the default value of 1.0×10^{-3}) is used to determine a scale factor. If neither the `SMALL STRAIN` command line nor the `VECTOR SCALE` command line appears in the `LANCZOS PARAMETERS` command block, then the small-strain value will be used to calculate the scale factor. If the `SMALL STRAIN` command line appears in the command block with any value other than the default value *and* the `VECTOR SCALE` command line does not appear in the command block, then the vector scale will be determined by the user-specified value for the small strain. If the `VECTOR SCALE` command line does appear in the `LANCZOS PARAMETERS` command block, then the vector scale will be determined by the user-specified value for the vector scale.

The recommended way of determining the scale factor is to use the small-strain approach with the default small-strain value. If you follow this advice, neither the `SMALL STRAIN` command line nor the `VECTOR SCALE` command line should be included in the `LANCZOS PARAMETERS` command block.

As indicated previously, the scale factor must not be too small, as this will create round-off problems and give a bad estimate for the critical time step. If the scale factor is too large, we violate

the above restriction of a constant tangent stiffness matrix.

The `SCALE FACTOR` command line sets the factor f_s in Equation (3.11). The value for f_s , i.e., `time_scale`, is set to 1.0.

The `UPDATE ON TIME STEP CHANGE` command line sets the value for Δt_{lim} in Equation (3.14). If the change in the element-based critical time step estimate as given by Equation (3.14) exceeds the value for Δt_{lim} , then the Lanczos method is called for a new estimate for the critical time step. The default value for Δt_{lim} , i.e., `tstep_change`, is 0.10.

The `UPDATE STEP INTERVAL` command line sets the number of step intervals at which the Lanczos method is called. If `step_int` is set to 1000, the Lanczos method will be called every one thousand steps to compute an estimate for the critical time step. (The default value of `step_int` is 500.) This control mechanism interacts with the control established by the `UPDATE ON TIME STEP CHANGE` command line. Suppose we have set `step_int` to 1000, and we have computed 800 steps since the last call to the Lanczos method. If Δt_{lim} has been set to 0.15 and we exceed this value at step 800, then the change in the element-based time step will result in the Lanczos method being called. The counter for keeping track of the number of step intervals since the last Lanczos computation will be reset to zero. The next call to the Lanczos method will occur again in one thousand steps, unless we again exceed the change in the element-based time step.

3.2.2 Power Method

The power method is a simple iterative technique that gives an estimate for the maximum eigenvalue. Consider the following form of the eigenvalue problem, which is useful for our explicit transient dynamics problem.

$$\mathbf{K}_T \mathbf{x}_i - \theta_i \mathbf{M} \mathbf{x}_i = 0 \quad (3.16)$$

Reformulate the problem as

$$[\mathbf{M}]^{-1} \mathbf{K}_T \mathbf{x}_i - \theta_i \mathbf{x}_i = 0. \quad (3.17)$$

Denote the matrix product $\mathbf{M}^{-1} \mathbf{K}_T$ as the matrix operator $\mathbf{A}$. The eigenvalue problem is now

$$\mathbf{A} \mathbf{x}_i - \theta_i \mathbf{x}_i = 0. \quad (3.18)$$

The maximum eigenvalue for the problem in Equation (3.18) can be computed with the following iterative process:

$$\begin{aligned} & \text{for } i = 1, n \\ & \quad \mathbf{x}_i = \mathbf{A} \mathbf{x}_{i-1} \\ & \quad \mathbf{x}_i = \mathbf{x}_i / \|\mathbf{x}_i\| \\ & \quad \theta_i = [\mathbf{x}_i]^T \mathbf{A} \mathbf{x}_i \\ & \text{end} \end{aligned}$$

The value θ_i is the i^{th} estimate for the maximum eigenvalue. As n increases, the power method should yield a more accurate estimate for the maximum eigenvalue. Similar to the Lanczos method, the power method must iterate at some given time step to obtain an estimate for the maximum

eigenvalue at that time step. The details for the above form of the power method are described in [1].

Like the Lanczos method, the power method requires the use of an internal force calculation. The power method, however, is somewhat simpler than the Lanczos method. The power method requires that one vector and one scalar be computed. The Lanczos method requires the computation of five vectors and two scalars. Further, the power method yields a direct estimate for the maximum eigenvalue, whereas the Lanczos method requires that the maximum eigenvalue be extracted from a tri-diagonal matrix.

If the sequence of eigenvalues for an analysis problem is

$$|\lambda_1| > |\lambda_2| \geq \dots \geq |\lambda_n|, \quad (3.19)$$

then the rate of convergence to the maximum eigenvalue depends on the ratio

$$\frac{|\lambda_1|}{|\lambda_2|}. \quad (3.20)$$

Depending on the magnitudes of $|\lambda_1|$ and $|\lambda_2|$, the rate of convergence could be rather slow. The power method, for a given problem, may not converge to as accurate an estimate for the maximum eigenvalue as quickly as the Lanczos method. We should be careful, however, in how we interpret this convergence property for the power method. For many of our problems, we probably have a large number of eigenvalues clustered slightly below the maximum eigenvalue. The power method may give a reasonable estimate for the maximum eigenvalue in a relatively small number of iterations. The power method may converge quickly to some eigenvalue estimate that is slightly below the maximum eigenvalue. Convergence to the maximum eigenvalue, in terms of accuracy to many digits, might then be very slow. However, one would still have an adequate estimate for the maximum eigenvalue to set a value for the critical time step.

One of the advantages of the power method is that we can use the last value for $\mathbf{x}_i$ from a previous call to the power method as a starting vector for our current call to the power method and converge quickly (for some problems) to an accurate estimate for the maximum eigenvalue. Consider the following scenario. We make an initial call to the power method using some arbitrary $\mathbf{x}_0$ starting vector. We obtain a good estimate for the maximum eigenvalue after n iterations. After some number of time steps, we again call the power method and we use $\mathbf{x}_n$ from the previous call to the power method as our current starting vector. Because we are solving nonlinear problems, the matrix operator, $\mathbf{A}$, has most likely changed from the time when we made the initial call to the power method. If the changes in the matrix operator are small, then $\mathbf{x}_n$ should be an excellent choice for the starting vector, and we should converge quickly to a good estimate for the maximum eigenvalue. This behavior, quick convergence to a good eigenvalue estimate using a prior value of $\mathbf{x}_n$ as a starting vector, has been observed in some sample problems. We discuss this issue of restarting the power method using a previous $\mathbf{x}_n$ in more detail in Section 3.2.2.1.

Obtaining a cost-effective and user-friendly implementation of the power method requires addressing the same issues discussed above for an automated implementation of the Lanczos method. First, we must determine some scheme to call the power method on an intermittent basis so that it is cost effective. The control mechanism for calling the Lanczos method on an intermittent basis can also be used for the power method. Second, the $\mathbf{x}$ vector must be scaled appropriately before

it is used in the internal force calculations. (For the Lanczos method, we must scale the vector $\mathbf{p}$.) Finally, we need to terminate the power method. A tolerance on the value in Equation (3.15) can be set to terminate the method. (Setting a tolerance on Equation (3.15) is the same approach used for the Lanczos method.)

3.2.2.1 Power Method with Constant Time Steps

The cost of making one iteration for the power method is approximately the cost of an internal force calculation. As indicated previously, the cost of one iteration for the Lanczos method (the calculation of a Lanczos vector) is also approximately the cost of an internal force calculation. Therefore, the equations developed in Section 3.2.1.1 are also applicable for determining when the power method can be used in a cost-effective manner. Recall that the equation

$$n_e = \frac{N_L}{1 - 1/r} \quad (3.21)$$

in Section 3.2.1.1 gives the number of time steps for the break-even point at which it becomes cost effective for the case of no contact to use the Lanczos method to reduce computational costs by overcoming the initial cost of the Lanczos calculations with the larger critical time step. In the above equation, N_L is the number of Lanczos iterations, and r is the ratio of the Lanczos-based critical time to the element-based time step estimate. This equation is directly applicable to the power method because the cost of one iteration in the power method is similar to the cost of one iteration in the Lanczos method. (The variable N_L becomes the number of iterations for the power method.)

Now let us return to the scenario in Section 3.2.2 where we look at restarting the power method with the last value for $\mathbf{x}_i$ from a previous call to the power method. We make an initial call to the power method using some arbitrary $\mathbf{x}_0$ starting vector. We obtain a good estimate for the maximum eigenvalue after n iterations. Our estimate for the critical time step based on the maximum eigenvalue gives us a value of 1.25 for r . After some number of time steps, we again call the power method, and we use $\mathbf{x}_n$ from the previous call to the power method as our current starting vector. In our second call to the power method, the power method converges to a good estimate for the maximum eigenvalue in two iterations. The break-even point where we recoup the cost of the power method (in our second call) is ten time steps. If the power method converges to a good estimate for the maximum eigenvalue in three iterations, then the break-even point where we recoup the cost of the power method (in our second call) is fifteen time steps. There are some problems where we can obtain a good estimate for the maximum eigenvalue in two or three iterations when we use the last value for $\mathbf{x}_n$ from a previous call to the power method as our starting vector for the current call to the power method. For the case where we can converge to a good estimate for the maximum eigenvalue in two or three iterations, we can easily call the power method once every forty or fifty time steps and use the power method to improve the time step estimate in a cost-effective manner. Over a time span of forty or fifty time steps, matrix operator $\mathbf{A}$, as defined in Equation (3.18) may change by only a small amount for a large class of problems. If matrix operator $\mathbf{A}$ changes by a small amount, then using the last value of $\mathbf{x}_n$ from a previous call to the power method provides an excellent starting vector and gives us quick convergence. This ability of the power method to converge quickly by using previous information makes it a viable scheme for calculating the maximum eigenvalue and producing a critical time step estimate.

For a problem with contact, the equations derived for the Lanczos method (with contact) in Section 3.2.1.1 are also applicable to the power method. For the power method and the Lanczos method, the added computational cost of the contact calculations results in reaching a break-even point with a smaller number of iterations when compared to a case where there is no contact.

3.2.2.2 Controls for Power Method

Using the power method is similar to using the Lanczos method in that we must be able to perform a power method calculation and reuse the power method estimate for the critical time step in some way over a number of subsequent time steps. The approach for reusing a power method estimate for the critical time step over a number of time steps is the same as the approach for reusing a Lanczos-based estimate over a number of time steps as described in Section 3.2.1.2. The approach discussed in Section 3.2.1.2 presents a way to reuse a maximum eigenvalue time step estimate over a number of time steps so that we maintain a critical time step estimate that is close to the theoretical maximum value in between the calls to the maximum eigenvalue calculations. The approach discussed in in Section 3.2.1.2 makes use of the element-based critical time step estimate at each time step.

3.2.2.3 Scale Factor for Power Method

When the power method is called for a given time step, it must appear that the calculations are using a constant matrix operator, $\mathbf{A}$, for all iterations. The $\mathbf{A}$ operator is the product of the inverse of the mass matrix $\mathbf{M}^{-1}$ and the tangent stiffness matrix $\mathbf{K}_T$ for the problem. The mass matrix is a constant. Therefore, we must assure that we are working with what is effectively a constant tangent stiffness matrix for all iterations during a call to the power method. This is the same requirement that we encounter in the Lanczos method (see Section 3.2.1.3). For the power method, we use the internal force calculations to generate the product $\mathbf{A}\mathbf{x}_i$ by first computing the internal forces using $\mathbf{x}_i$, which gives us the product $\mathbf{K}_T\mathbf{x}_i$, and then multiplying the internal force results by the inverse of the mass matrix. Any vector $\mathbf{x}_i$, as calculated by the power method, may be such that it represents large-strain behavior and moves the internal force calculations into a nonlinear regime. It is necessary to scale the $\mathbf{x}_i$ vectors so that the internal force calculations are in a small-strain regime, which makes it appear that we are working with a constant tangent stiffness matrix. The vectors $\mathbf{x}_i$ must be scaled so that they represent velocities associated with small-strain calculations. When properly scaled vectors are sent to the internal force calculation, the internal force calculation becomes a matrix $\times$ vector product with a constant tangent stiffness matrix for all iterations during a given call to the power method.

The scale factor for the $\mathbf{x}_i$ vectors, which we will designate as v_{sf} , must not be too small, as this will create round-off problems and give a bad estimate for the critical time step. If the scale factor is too large, we violate the above restriction of a constant tangent stiffness matrix.

There are two approaches for controlling the scale factor when the power method is used to compute the maximum eigenvalue. These approaches are discussed in Section 3.2.2.5.

3.2.2.4 Accuracy of Eigenvalue Estimate

To use the power method efficiently, it is best not to specify a set number of iterations for every instance when the power method is called. Instead, some tolerance should be set on the convergence measure defined in Equation (3.15). The initial call to the power method may take a significant number of iterations to produce a good estimate for the maximum eigenvalue. On subsequent calls to the power method, the number of iterations to converge to a good estimate for the maximum eigenvalue should be small when we use previous values of $\mathbf{x}$ as a starting vector.

For the power method, we rely on the convergence measure defined by Equation (3.15) to control the number of iterations. The user can set an upper limit on the number of iterations for the power method. Control of the number of iterations for the power method is discussed in more detail in Section 3.2.2.5.

3.2.2.5 Power Method Parameters Command Block

```
BEGIN POWER METHOD PARAMETERS <string>powermethod_name
  STARTING VECTOR = <string>STRETCH_X|STRETCH_Y|STRETCH_Z|
    ISOTHERMAL(ISOTHERMAL)
  INCREASE OVER STEPS = <integer>incr_int(5)
  NUMBER ITERATIONS = <integer>num_iter(150)
  EIGENVALUE CONVERGENCE TOLERANCE =
    <real>converge_tol(0.5e-3)
  SMALL STRAIN = <real>small_strain(1.0e-3)
  VECTOR SCALE = <real>vec_scale(1.0e-5)
  SCALE FACTOR = <real>time_scale(0.9)
  UPDATE ON TIME STEP CHANGE = <real>tstep_change(0.10)
  UPDATE STEP INTERVAL = <integer>step_int(50)
END [POWER METHOD PARAMETERS <string>powermethod_name]
```

If you use the power method to compute a critical time step, there should be only one `POWER METHOD PARAMETERS` command block, and it should appear in the region. If you have a `POWER METHOD PARAMETERS` command block, you should not specify a command block for the node-based method or the Lanczos method. If you use the power method to compute the critical time step, the time step increase factor (default or user-specified) set in the `TIME CONTROL` command block will be used to control the increase in the time step estimate; the time step scale factor (default or user-specified) set in the `TIME CONTROL` command block will not be used. These factors are specified in the `PARAMETERS FOR PRESTO REGION` portion of a `TIME STEPPING BLOCK` in the `TIME CONTROL` command block.

The power method requires some type of starting vector. This is determined by the `STARTING VECTOR` command line. The various options available in this command line will generate a displacement vector that stretches your model in the x -, y -, or z -direction, or in all three directions (x , y , and z) at once. (`STRETCH_X` stretches the model in the x -direction, `STRETCH_Y` stretches the model in the y -direction, `STRETCH_Z` stretches the model in the z -direction, and `ISOTHERMAL` stretches the model in all three directions at once.) The displacement vector then serves as a basis for generating the starting vector x_0 in the initialization phase of the Lanczos method. The default starting vector is the `ISOTHERMAL` option. You may encounter cases where use of one of the other

options for a starting vector—`STRETCH_X`, `STRETCH_Y`, or `STRETCH_Z`—may result in a slight increase in accuracy for the critical time step estimate for a given number of iterations. These other options (`STRETCH_X`, `STRETCH_Y`, and `STRETCH_Z`) are offered for these special cases.

The `INCREASE OVER STEPS` command line determines how many steps, `incr_int`, are used to transition from an element-based critical time step estimate to the power method estimate at the beginning of an analysis. The user may want to increase from the element-based estimate to the power method estimate over a number of time steps if the difference between these two estimates is large. The value of `incr_int` defaults to 5.

As indicated in Section 3.2.2.4, some number of iterations are required by the power method to obtain a good estimate for the maximum eigenvalue for a given model. The number of iterations is controlled by the tolerance set for the convergence measure given in Equation (3.15). The tolerance is set with the `EIGENVALUE CONVERGENCE TOLERANCE` command line. The default value for the convergence tolerance, `converge_tol` is 0.5×10^{-3} . (This default value may change in the future as we gain more experience with the power method.)

The user can set an upper limit on the number of iterations for the power method by using the `NUMBER ITERATIONS` command line. If the number of iterations in the power method exceeds the value set by `num_iter`, then the power method is terminated. Otherwise, the iterative process in the power method is terminated by the tolerance set on the convergence measure. The default value for `num_iter` is 150.

As indicated in Section 3.2.2.3, it is necessary to scale the vectors $\mathbf{x}_i$ (with a scale factor v_{sf}) in the power method calculations so that it appears we are always working with a constant tangent stiffness each time we call the power method. There are two approaches for determining the scale factor. With the first approach, the user can specify the value of a small strain for a particular model by using the `SMALL STRAIN` command line. (As long as the internal force calculation inside the power method is working with a vector that approximates small-strain behavior, it will appear that we are working with a constant tangent stiffness inside the power method.) The value for v_{sf} is then computed based on your specification for what constitutes a small strain in the model. (The default value for `small_strain` is 1.0×10^{-3} .) With the second approach for determining the scale factor, the user can specify a value for v_{sf} directly by using the `VECTOR SCALE` command line. The default value for v_{sf} , the `vec_scale` parameter in that command line, is 1.0×10^{-5} . A number of tests have established that this value for the scale factor, 1.0×10^{-5} , works well for a range of models encountered at Sandia National Laboratories. If the user wants to use the option of directly specifying the scale factor, it is possible to determine whether a particular scale factor is suitable for a particular problem. Take a scale factor v_{sf} , plus values on either side of it, say, $0.9 \times v_{sf}$ and $1.1 \times v_{sf}$. If all three of these scale factor values produce almost the same estimate for the critical time step for a particular model, then the value for v_{sf} meets the criterion for an acceptable scale factor.

By default, the small-strain approach (with the default value of 1.0×10^{-3}) is used to determine a scale factor. If neither the `SMALL STRAIN` command line nor the `VECTOR SCALE` command line appears in the `POWER METHOD PARAMETERS` command block, then the default small-strain value will be used to calculate the scale factor. If the `SMALL STRAIN` command appears in the command block with any value other than the default value *and* the `VECTOR SCALE` command line does not appear in the command block, then the vector scale will be determined by the user-specified value

for the small strain. If the `VECTOR SCALE` command line does appear in the `POWER METHOD PARAMETERS` command block, then the vector scale will be determined by the user-specified value for the vector scale.

The recommended way of determining the scale factor is to use the small-strain approach with the default small-strain value. Thus, if you follow this advice, neither the `SMALL STRAIN` command line nor the `VECTOR SCALE` command line should be included in the `POWER METHOD PARAMETERS` command block.

As indicated previously, the scale factor must not be too small, as this will create round-off problems and give a bad estimate for the critical time step. If the scale factor is too large, we violate the above restriction of a constant tangent stiffness matrix.

The `SCALE FACTOR` command line sets the factor f_s in Equation (3.11). The value for f_s , i.e., `time_scale` is set to 0.9. More experience with the power method will help us determine whether a slightly larger value of f_s can be used as a default.

The `UPDATE ON TIME STEP CHANGE` command line sets the value for Δt_{lim} in Equation (3.14). If the change in the element-based critical time step estimate as given by Equation (3.14) exceeds the value for Δt_{lim} , then the power method is called for a new estimate for the critical time step. The default value for Δt_{lim} , i.e., `tstep_change`, is 0.10.

The `UPDATE STEP INTERVAL` command line sets the number of step intervals at which the power method is called. If `step_int` is set to 40, the power method will be called every forty steps to compute an estimate for the critical time step. (The default value of `step_int` is 50.) This control mechanism interacts with the control established by the `UPDATE ON TIME STEP CHANGE` command line. Suppose we have set `step_int` to 40, and we have computed thirty-five steps since the last call to the power method. If Δt_{lim} has been set to 0.15 and we exceed this value at step 35, then the change in the element-based time step will result in the power method being called. The counter for keeping track of the number of step intervals since the last power method computation will be reset to zero. The next call to the power method will then occur again in forty steps, unless we again exceed the change in the element-based time step.

3.2.3 Node-Based Method

Now that we have developed schemes to make the Lanczos method and power method cost-effective tools for estimation of the critical time step, let us examine the node-based scheme.

The node-based method in Presto will give an estimate for the critical time step that is greater than or equal to the element-based estimate. It may or may not give an estimate for the time step that is close to the maximum value associated with the maximum eigenvalue for the problem. In general, we assume the node-based method will give us an estimate larger than the element-based estimate, but not significantly larger.

If the node-based scheme is used to determine the critical time step for a block of uniform elements (all the same material), then the estimate from the node-based method will be the same as that for the element-based estimate. The node-based estimate begins to diverge from (become larger than) the element-based estimate as the differences in aspect ratios of the elements attached to a node become larger. We can assume, therefore, that if the stiffest part of our structure has a relatively

uniform mesh, the node-based method will not give a significantly larger estimate than the element-based method.

The node-based method costs only a fraction of an internal force calculation. However, the node-based method makes use of element time step estimates. Therefore, to use the node-based method, one must also do the element-based method. The cost of the node-based method is in addition to the element-based method. The modest increase in the critical step estimate from the node-based method is unlikely to offset the added cost of the node-based method.

To make the node-based method cost-effective, we can use the same methodology that is employed for the Lanczos method. Let t_b be the estimate for the critical time step from the node-based method. We define the ratio of the node-based estimate to the element-based estimate as

$$t_r = \frac{\Delta t_b}{\Delta t_e}. \quad (3.22)$$

This ratio is then used to scale subsequent element-based estimates for the critical time step. If $\Delta t_{e(n)}$ is the n^{th} element-based critical time step after the time step where the node-based calculations are performed, then the n^{th} time step after the node-based calculations, $\Delta t_{(n)}$, is simply

$$\Delta t_{(n)} = t_r \Delta t_{e(n)}. \quad (3.23)$$

The ratio t_r is used until the next call to the node-based method. The next call to the node-based method is controlled by one of two mechanisms described in the Lanczos discussion. The node-based method is called after a set number of times or a significant change in the element-based estimate for the critical time step.

3.2.3.1 Node-Based Parameters Command Block

```
BEGIN NODE BASED TIME STEP PARAMETERS <string>nbased_name
  INCREMENT INTERVAL = <integer>incr_int(0)
  STEP INTERVAL = <integer>step_int(1)
  TIME STEP LIMIT = <real>step_lim(0.10)
END [NODE BASED TIME STEP PARAMETERS <string>nbased_name]
```

If you use the node-based method to compute a critical time step, there should be only one NODE BASED TIME STEP PARAMETERS command block, and it should appear in the region. If you use the node-based method to compute the critical time step, you should set the time step scale factor to 1.0 and the time step increase factor to a large number (5.0 is an acceptable value for the time step increase factor). These factors are specified in the PARAMETERS FOR PRESTO REGION portion of a TIME STEPPING BLOCK in the TIME CONTROL command block. If you have a NODE BASED TIME STEP PARAMETERS command block, you should not specify a LANCZOS PARAMETERS command block.

The INCREMENT INTERVAL command line determines how many steps, `incr_int`, are used to transition from an element-based critical time step estimate to the node-based estimate at the beginning of an analysis. The user may want to increase from the element-based estimate to the node-based estimate over a number of time steps if the difference between these two estimates is

large. The value of `incr_int` defaults to 0 so the node based time step is used immediately at the beginning of the analysis.

The `STEP INTERVAL` command line sets the number of step intervals at which the node-based method is called. If `step_int` is set to 1000, the node-based method will be called every one thousand steps to compute an estimate for the critical time step. (The default value of `step_int` is 1.) Increasing the step interval can improve performance, but depending on the problem could run the risk of sending the time step super-critical. This control mechanism interacts with the control established by the `TIME STEP LIMIT` command line. Suppose we have set `step_int` to 1000, and we have computed 800 steps since the last call to the node-based method. If Δt_{lim} has been set to 0.15 and we exceed this value at step 800, then the change in the element-based time step will result in the node-based method being called. The counter for keeping track of the number of step intervals since the last node-based computation will be reset to zero. The next call to the node-based method will occur again in one thousand steps, unless we again exceed the change in the element-based time step.

The `TIME STEP LIMIT` command line sets the value for Δt_{lim} ,

$$\Delta t_{lim} = \frac{|\Delta t_{e(n)} - \Delta t_e|}{\Delta t_e} \quad (3.24)$$

This is exactly the same as Equation (3.14) presented eleven pages earlier. If the change in the element-based critical time step estimate Δt_{lim} , exceeds the value for Δt_{lim} , then the node-based method is called for a new estimate for the critical time step. The default value for Δt_{lim} , i.e., `step_lim`, is 0.10.

3.3 Mass Scaling

3.3.1 What is Mass Scaling?

Mass scaling allows for arbitrarily increasing the mass of certain nodes to increase the global estimate of the critical time step; therefore, mass scaling is used in conjunction with a node-based time step. The nodes where the mass is increased must be associated with elements that control the global time step; increasing the mass at a node of an element effectively raises the critical time step estimate for that element.

Mass scaling only adjusts the *mass* at the nodes. It does not, however, modify the *density* value used in the element calculations, even though the net effect of mass scaling makes it appear as though it has.

Mass scaling can be useful in a number of circumstances, such as

- quasi-static or rigid-body motion: If the model or part of a model is undergoing what is nearly quasi-static or rigid-body motion, adding mass may have little effect on the end result.
- disparate element sizes: A model may contain groups of elements that are much smaller than others. For example, in screws or gears that are modeled in detail, elements in the screw threads or gear teeth could be much smaller than elements in other portions of the model. If the dynamic response of these small parts is unimportant relative to the rest of the model, adding mass to them might not affect the quantities of interest.
- increasing the critical time step for “unimportant” model components: In some problems, a relatively unimportant mesh feature may control the critical time step. Consider, for example, a car crash simulation in which the bumper is the first part to strike an object. The crumpling deformation could reduce the time step in some elements in the bumper, potentially controlling the global critical time step. At a later time in the analysis, however, the effect of the bumper on the overall crash dynamics may not be significant; thus, mass scaling could be applied to ensure that the noncritical bumper does not control the global time step.



Warning: Use of mass scaling *will* introduce unbounded and potentially unpredictable errors into the analysis and should thus be used with caution. The analyst must use discretion to determine whether mass scaling may be used in a way that does not distort the results of interest.

3.3.2 Mass Scaling Command Block

```
BEGIN MASS SCALING
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
ELEMENT = <string list>element_names
NODE = <string list>element_names
INCLUDE ALL BLOCKS
```

```

REMOVE NODE SET = <string list> nodelist_names
REMOVE SURFACE = <string list> surface_names
REMOVE BLOCK = <string list> block_names
#
TARGET TIME STEP = <real> target_time_step
ALLOWABLE MASS INCREASE RATIO = <real> mass_increase_ratio
#
# additional command
ACTIVE PERIODS = <string list> periods
INACTIVE PERIODS = <string list> periods

UPDATE INTERVAL = INITIALIZATION_ONLY | EVERY_STEP | AUTOMATIC
(AUTOMATIC)
END MASS SCALING

```

The `MASS SCALING` command block controls mass scaling for a specified set of nodes. This command block contains one or more command lines to specify the node set. It also contains two command lines that determine how the actual mass scaling will be applied to the nodes in the node set. In addition to the command lines in the two command groups, there are two additional command lines: `ACTIVE PERIODS` and `INACTIVE PERIODS`. These command lines are used to activate or deactivate the mass scaling for specified time periods.

Multiple `MASS SCALING` command blocks can exist to apply different criteria to different portions of the mesh at different times. For any given set of `MASS SCALING` command blocks, mass will only be added to a node if doing so will allow increasing the global time step. The amount of artificial mass added to a node will vary in time as the mesh deforms and moves. The added mass computation is recomputed every time the node-based time step estimate is recomputed.

NOTE: Mass scaling *must* be used in conjunction with the node-based time step estimation method. Consult Section 3.2.3 for a description of the node-based method for estimating the critical time step.

Following are descriptions of the different command groups:

3.3.3 Node Set Commands

The `node set commands` portion of the `MASS SCALING` command block specifies the nodes associated with the boundary condition. This portion of the command block can include some combination of the following lines:

```

NODE SET = <string list> nodelist_names
SURFACE = <string list> surface_names
BLOCK = <string list> block_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list> nodelist_names
REMOVE SURFACE = <string list> surface_names
REMOVE BLOCK = <string list> block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes for mass scaling. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` command line in the command block.

3.3.3.1 Mass Scaling Commands

The `MASS SCALING` command block may contain either a

```
TARGET TIME STEP = <real>target_time_step
```

command line or an

```
ALLOWABLE MASS INCREASE RATIO = <real>mass_increase_ratio
```

command line, or both of these command lines can appear in the input block.

The `TARGET TIME STEP` command line sets the maximum time step for a set of nodes. The parameter `target_time_step` is the maximum time step for all the nodes specified in the command block.

The `ALLOWABLE MASS INCREASE RATIO` command line sets an upper limit on the mass scaling at a node. The value specified for `mass_increase_ratio` limits the ratio of the mass at a node, as set by mass scaling, to the original mass at the node. (The original mass of the node is determined only by the element contributions.) This ratio must be a factor greater than or equal to 1. If m_s is the scaled mass at a node and m_0 is the original mass at the node due only to element contributions, then the ratio $m_s \div m_0$ will not exceed the value of `mass_increase_ratio`.

Mass scaling will add mass to nodes until the target time step is reached, the mass added to some node reaches the allowable mass increase ratio, or the current set of nodes no longer controls the global analysis time step.

The amount of mass added due to mass scaling is stored in the nodal variable `mass_scaling_added_mass`. This variable can be output and post-processed to determine how much mass is being added at a given time. See Section 9.3.1 regarding the output of nodal variables to a results file.

3.3.3.2 Additional Commands

The `ACTIVE PERIODS` or `INACTIVE PERIODS` command lines can optionally appear in the `MASS SCALING` command block:

```
ACTIVE PERIODS = <string list>periods
INACTIVE PERIODS = <string list>periods
```

These command lines determine when mass scaling is active. See Section 2.6 for more information about these command lines.

The `UPDATE INTERVAL` command defines how often mass scaling is updated. The default `AUTOMATIC` updates the mass scaling when the time step has changed significantly (greater than 5% of a frequency). The `EVERY_STEP` option recalculates the mass scaling every single step based off of the current model stiffness. Use of this option will ensure the model run at exactly

the specified time step. The `INITIALIZATION_ONLY` option will calculate the mass scaling once at the first time step. The nodal mass scaling will not change after this point. This option can be useful to accept and bound how much mass will be added to the model once and not have the possibility of the mass changing later in time.

References

- [1] G.H. Golub and C.F. Van Loan. *Matrix Computations, Third Edition*. The John Hopkins University Press, Baltimore, MD, 1996.
- [2] L.N. Trefethen and D. Bau. *Numerical Linear Algebra*. SIAM, 1997.
- [3] T.J.R. Hughes. *The Finite Element Method—Linear Static and Dynamic Finite Element Analysis*. Prentice-Hall, Inc, Englewood Cliffs, NJ, 1987.



Chapter 4

Implicit Solver and Time Step Control

This chapter discusses the commands used to control the solver, time stepping, and dynamics parameters for implicit calculations. The commands and features documented in this chapter are applicable only for quasistatic and implicit dynamic analyses. For information on time stepping commands for explicit dynamic analyses, see Chapter 3.

The solution strategy used in Sierra/SM is based on iterative solution techniques and builds on the capabilities pioneered in the JAS3D code [1]. Sierra/SM uses a nonlinear preconditioned conjugate gradient (CG) algorithm [2] to iteratively find a solution that satisfies equilibrium within a user-specified error tolerance at each load step.

The nonlinear CG algorithm iterates to a solution by computing a series of search direction vectors that are successively added to the trial velocity vector. In each iteration, the residual is multiplied by a preconditioning matrix to obtain a gradient direction. The gradient direction is then used to compute the next search direction in a way that ensures that each new search direction is orthogonal to the previous search directions. A line search is used to compute a scaling factor that when applied to the search direction results in a minimized residual.

The performance of the CG algorithm is highly dependent on the choice of preconditioner. Using the inverse of the full tangent stiffness matrix as a preconditioner minimizes the number of CG iterations required but can be computationally expensive. Alternatively, the inverse of a diagonalized stiffness matrix can be used. This requires much lower computational and memory resources, per iteration, but may require many more iterations.

Sierra/SM provides several preconditioner options that can be categorized in two general types. The first type is the nodal preconditioner. Several different variants are available including the diagonal preconditioners that were available in JAS3D. The second type is the full tangent preconditioner. This option is typically implemented through use of the parallel-scalable FETI linear solver.

Conditioning of a problem can also be improved by using a multilevel solver. Rather than directly solving the difficult, and potentially ill-posed, full problem with the CG solver, Sierra/SM forms a series of “model problems” and solves them with the core CG solver. Features of the model that make it difficult are either adjusted or held constant for a model problem. After each model problem is solved, the multilevel solver updates the quantities that were controlled. The process of forming model problems and updating is iterative, as with the core solver, and convergence must be achieved both in the core solver and in the multilevel controls.

Four types of controls are available for the multilevel solver. They are discussed at a high level in

the next section.

This chapter discusses the commands for specifying solver options for both quasistatic and implicit dynamic problems. The CG solver can either be used alone or as the core solver within a multilevel solver. Section 4.1 presents the characteristics and structure of the `SOLVER` command block, which contains all solver-related commands within the `ADAGIO REGION` command block. Section 4.2 discusses the `CG` command block, which controls the nonlinear preconditioned CG solver. An option in the `CG` command block is to use a full tangent preconditioner. The command block to control the behavior of this option is discussed in Section 4.3. The recommended linear solver for use with the full tangent preconditioner is FETI, whose command block is presented in Section 4.4. Section 4.9 describes the predictors, which are used to generate an initial trial solution for a load step or for a multilevel-solver model problem. Time stepping control for both quasistatic (where time serves as a pseudo-time load step parameter) and implicit dynamic problems is described in Section 4.10. Section 4.11 discusses techniques to perform implicit dynamic solutions. Finally, references are provided at the end of the chapter.

4.1 Multilevel Solver

Sierra/SM allows the conjugate gradient (CG) solver to be used either by itself or as the core solver within a multilevel solver. The multilevel solver improves the ability of the core CG solver to solve poorly conditioned problems by holding fixed some variables that would ordinarily be free to change in a full nonlinear iteration. With a multilevel solver CG iterations are performed while a variable is held fixed. After convergence is obtained with the controlled variable fixed, a new residual calculation is performed, and the controlled variable is updated. Moreover, the multilevel solver algorithms can be used in parallel and/or layered to treat multiple variables by assigning them to multiple controls. Convergence on any particular level requires that all levels leading up to that level be converged. Some Sierra/SM capabilities can be used only in conjunction with the appropriate multilevel solver. For example, a contact problem generally requires a control contact solver level, and a problem modeling material fracture with element death requires a control failure solver level.

As an example, consider the case illustrated in Figure 4.1 of a solver with two controls defined, one at a level above the other. Once all control variables have been initially established for a load step, CG iterations are performed until convergence is obtained on the core “model problem.” At this point, the first variable controlled above the CG solve, also known as the “level 1” control, is adjusted to minimize its residual. This process is known as a “level 1 update.” After the level 1 update, the model problem is solved again, and this process is repeated until the core solver and the variable controlled at level 1 have converged. Once a solution has been obtained at level 1, the variable controlled at level 2 is updated. After each level 2 update, the level 1 problem must be solved again. For convergence in the multilevel solver, all updates starting with the lowest level and proceeding to the highest level must be in equilibrium. Sierra/SM allows up to two control levels, and multiple controls can be defined at each level (though not with control failure, see Section 4.7).

Sierra/SM has four types of controls that can be used within the multilevel solver: control contact, control stiffness, control failure, and control damped solve.

Control contact is used to keep the set of nodes in contact constant during the solution of a model problem. Contact searches are performed during the multilevel solver updates. Control contact must be used for all problems that include a contact definition block, with the one exception of kinematic tied contact. If no control contact block is defined for problems that require one, a default control contact will be used. Control contact is not required for models that have only kinematic constraints, such as kinematic tied contact and MPCs. Control contact is documented in detail in Section 4.5.

Control stiffness is used to improve the conditioning of models that contain widely varying stiffness in different modes of material response. Such differences naturally exist for models with nearly incompressible materials where the bulk behavior is much stiffer than the shear behavior, for models with orthotropic and/or anisotropic materials that have much higher stiffness in preferred material directions, and for models containing several materials that are vastly different in their overall stiffness. Control stiffness works by scaling a given material response up (stiffening) or down (softening) to create a set of better conditioned model problems. For nearly incompressible materials, for instance, the bulk stress increments may be softened and/or the shear stress increments may be stiffened. For oriented materials, the stress increments in certain material directions are softened to

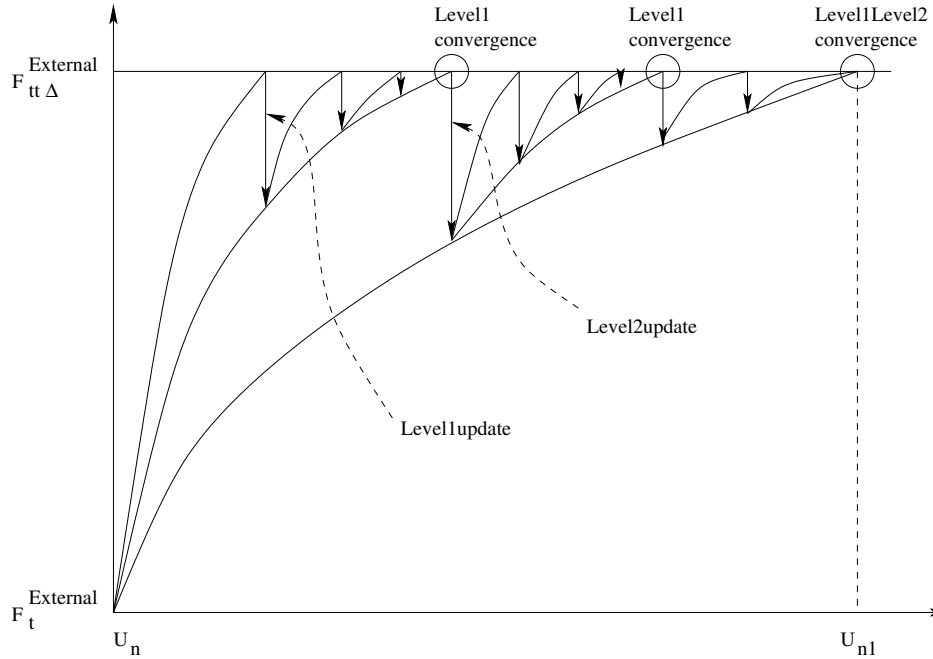


Figure 4.1: Reaching convergence with controls and the multilevel solver.

create a material response that is more isotropic in nature. Finally, for the case where a material is much stiffer than the surrounding materials, both the bulk and shear stress increments are softened to create a response that has a stiffness much closer to that of the adjacent materials. Ultimately, as the control stiffness model problems progress, the true material response is recovered by building up stresses that correspond to the true material response. Special material models are required to use this capability, and Sierra/SM provides a number of material models that are designed to work with this capability. Control stiffness is documented in detail in Section 4.6.

Control failure is used to improve the conditioning of models that involve element death due to failure defined within a material model or to keep a consistent state for other failure criteria. Control failure used with material criterion element death limits the failure only to the single most critical element during each control failure iteration. Iterations of control failure continue until no additional elements are marked for failure. Control failure is documented in detail in Section 4.7.

Control damped solve is used on problems that have initial rigid body modes or other problems that are inherently difficult to solve with a quasistatic implicit solver alone. Some examples of problems that may benefit from this control are problems that require contact to be seated prior to preload, implicit contact problems where rebound or bounce occur, difficult implicit contact problems in general, and problems with softening materials. Control damped solve is documented in detail in Section 4.8.

The following is the basic structure of the SOLVER command block:

```
BEGIN SOLVER

# CG solver commands:
```

```

BEGIN CG
  # ... see Sections 4.2, 4.3, and 4.4
END [CG]

# Multilevel control commands:

BEGIN CONTROL CONTACT [<string>contact_name]
  # ... see Section 4.5
END [CONTROL CONTACT <string>contact_name]

BEGIN CONTROL STIFFNESS [<string>stiffness_name]
  # ... see Section 4.6
END [CONTROL STIFFNESS <string>stiffness_name]

BEGIN CONTROL FAILURE [<string>failure_name]
  # ... see Section 4.7
END [CONTROL FAILURE <string>failure_name]

BEGIN CONTROL DAMPED SOLVE [<string>ds_name]
  # ... see Section 4.8
END [CONTROL DAMPED SOLVE <string>ds_name]

# Predictor commands:

BEGIN LOADSTEP PREDICTOR
  # ... see Section 4.9.1
END [LOADSTEP PREDICTOR]

LEVEL 1 PREDICTOR = <string>NONE|DEFAULT(DEFAULT)
# see Section 4.9.2

END [SOLVER]

```

The **SOLVER** command block contains all solver-related commands, and must be present in the **ADAGIO REGION** command block. It is used regardless of whether the **CG** solver should be used alone or as a core solver in the multilevel solver. The **CG** command block (described in Section 4.2) is required for all models. If no multilevel controls are desired, the commands related to those controls are simply omitted from the **SOLVER** block.

In addition to the **CG** command block, the **SOLVER** command block contains command blocks that describe the controls used.

Aside from the command blocks for the core solver and the controls, the **SOLVER** block contains the **LOADSTEP PREDICTOR** command block, which is described in Section 4.9.1, and the **LEVEL 1 PREDICTOR** command line, which is described in Section 4.9.2.

Convergence tolerances for the **CONTROL CONTACT**, **CONTROL STIFFNESS**, and **CONTROL DAMPED SOLVE** solver levels are set within those command blocks while the tolerance of the core

CG solver is set within the CG command block. Rather than using a convergence tolerance, the CONTROL FAILURE block is considered converged when no new elements are marked for failure during an iteration or when the maximum iterations is reached.

The commands within the CG command block are used in the same way whether the multilevel solver is used. It is, however, important to consider the role of tolerances within the multilevel solver. The convergence of the problem is controlled by the convergence of the highest solver level. For the multilevel solver to converge well, it is important for the core solver (or lower level of the solver, in the case of multiple levels) to reduce the residual by a certain amount during each model problem solution. For this reason, it is recommended that the solution tolerances be set an order of magnitude tighter for each lower level solver. The MINIMUM RESIDUAL IMPROVEMENT command in the CG command block is helpful for ensuring that the residual is reduced by the core solver during model problems, even though the residual might already be within the convergence tolerances (see Section 4.2.1).

4.1.1 Common Multilevel Solver Commands for All Controls

```
BEGIN SOLVER

# CG solver commands:

BEGIN CG

# Parameters for CG
# ... see Sections 4.2, 4.3, and 4.4

# Diagnostic output commands, off by default:

ITERATION PLOT = <integer>iter_plot
    [DURING <string list>period_names]
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks

END [CG]

#General Control commands:

BEGIN CONTROL CONTACT|STIFFNESS|FAILURE|{DAMPED SOLVE} \#
    [<string>control_name]

# ... see Sections 4.5, 4.6, 4.7, and 4.8

# Level selection command:

LEVEL = <integer>control_level

# Diagnostic output commands, off by default:

ITERATION PLOT = <integer>iter_plot
```

```

[DURING <string list>period_names]
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks

END [CONTROL CONTACT|STIFFNESS|FAILURE|{DAMPED SOLVE} \#
    [<string>control_name]]

END [SOLVER]

```

Displayed in this section are common commands to all controls that can be used for the multilevel solver. Other commands may be used for specific controls, but those commands are not listed in this section. For commands specific to `CONTROL CONTACT`, see section [Sections 4.5](#). For commands specific to `CONTROL STIFFNESS`, see section [4.6](#). For commands specific to `CONTROL FAILURE`, see section [4.7](#). For commands specific to `CONTROL DAMPED SOLVE`, see section [4.8](#).

```
LEVEL = <integer>control_level(1)
```

The `LEVEL` command line is used to specify the level in the multilevel solver at which contact, stiffness, and/or failure is controlled. The `level` parameter can have a value of either 1 or 2, with 1 being the default for all controls.

This command is used when multiple controls (e.g., control contact and control stiffness) are active in the multilevel solver. It is permissible for multiple controls to exist at a given level. In order to have a control at level 2, another control at level 1 is required. If control failure is defined, it should be the only control at the outermost (highest numerical value) level.

```

ITERATION PLOT = <integer>iter_plot
[DURING <string list>period_names]
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks

```

The iteration plot command lines can be used to output diagnostic information for each of the controls. The `ITERATION PLOT` command line allows plots of the current state of the model to be written to the output database during the control updates at a frequency specified by the value of `iter_plot`. The plots are output with fictitious time steps bounded by the step limits. Every time such a plot is generated, a message is sent to the log file. By default no iteration plots are generated. This feature is useful for debugging but should be used with care to avoid excessive output.

The output produced by iteration plots is sent by default to every active results output file specified by a `RESULTS OUTPUT` command block. The `ITERATION PLOT OUTPUT BLOCKS` command line can optionally be used to supply a list of output command block names to which iteration plots should be written. Each of the output block names listed in `plot_blocks` must match the name of a `RESULTS OUTPUT` command block (see [Section 9.3.1](#)). The `ITERATION PLOT OUTPUT BLOCKS` command line can be useful for directing iteration plots to separate output files so that these plots are separated from the converged solution output. It can also be used to send output plots for different levels of the multilevel solver to different files.

4.2 Conjugate Gradient Solver

The core nonlinear preconditioned conjugate gradient solver is controlled through the `CG` command block. This must be nested within a `SOLVER` command block, regardless of whether multilevel controls are used or not.

```
BEGIN CG
#
# convergence commands
# Default is not defined.
TARGET RESIDUAL = <real>target_resid
    [DURING <string list>period_names]
TARGET RELATIVE RESIDUAL = <real>target_rel_resid(1.0e-4)
    [DURING <string list>period_names]
# Default is 10 times target residual
# or 1e20 if a multilevel solver is defined
# and control failure is not defined.
ACCEPTABLE RESIDUAL = <real>accept_resid
    [DURING <string list>period_names]
# Default is 10 times target relative residual
# or 1e20 if a multilevel solver is defined
# and control failure is not defined.
ACCEPTABLE RELATIVE RESIDUAL = <real>accept_rel_resid
    [DURING <string list>period_names]
# Default is not defined.
REFERENCE = EXTERNAL|INTERNAL|BELYTSCHKO|RESIDUAL|ENERGY
    (EXTERNAL) [DURING <string list>period_names]
RESIDUAL NORM TYPE = ALL|TRANSLATION|SCALE_RB_ROTATIONS
    (ALL) [DURING <string list>period_names]
MINIMUM RESIDUAL IMPROVEMENT = <real>resid_improvement
    [DURING <string list>period_names]
MINIMUM ITERATIONS = <integer>min_iter(0)
    [DURING <string list>period_names]
RESIDUAL ROUNDOFF TOLERANCE = <real>(1.0e-15)
#
# default = 100 for tangent preconditioner.
# default = max(NumNodes,1000) for all other preconditioners.
MAXIMUM ITERATIONS = <integer>max_iter
    [DURING <string list>period_names]
#
# preconditioner commands
PRECONDITIONER = BLOCK|BLOCK_INITIAL|DIAGONAL|
    ELASTIC|IDENTITY|PROBE|TANGENT|
    DOMAIN (ELASTIC) [<real>scaling_factor]
PRECONDITIONER ITERATION UPDATE = <integer>iter_update
BALANCE PROBE = <integer>balance_probe(1)
    [DURING <string list>period_names]
NODAL PROBE FACTOR = <real>probe_factor(1.0e-6)
```



```

    [DURING <string list>period_names]
NODAL DIAGONAL SCALE = <real>nodal_diag_scale(0.0)
    [DURING <string list>period_names]
NODAL DIAGONAL SHIFT = <real>nodal_diag_shift(0.0)
    [DURING <string list>period_names]
BEGIN FULL TANGENT PRECONDITIONER
    # ... see Section 4.3
END [FULL TANGENT PRECONDITIONER]
#
# line search command, default is secant
LINE SEARCH ACTUAL|TANGENT [DURING <string list>period_names]
LINE SEARCH SECANT [<real>scale_factor]
LINE SEARCH BACKTRACK [<real>min_step(1.0e-3)
                        <real>reduction(0.95)
                        <real>init_step(1.0)]
LINE SEARCH TYPE MULTI = <real>num_domain(2)
    [DURING <string list>period_names]
#
# diagnostic output commands
ITERATION PRINT = <integer>iprint(25)
    [DURING <string list>period_names]
# off by default.
ITERATION PLOT = <integer>iplot
    [DURING <string list>period_names]
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks
#
# cg algorithm commands
ITERATION RESET = <integer>iter_reset(infinity)
    [DURING <string list>period_names]
ORTHOGONALITY MEASURE FOR RESET = <real>ortho_reset(0.5)
    [DURING <string list>period_names]
RESET LIMITS <integer>iter_start <integer>iter_reset
    <real>reset_growth <real>reset_orthogonality
    [DURING <string list>period_names]
BETA METHOD = FletcherReeves|PolakRibiere|Newton|
    PolakRibierePlus(PolakRibiere)
    [DURING <string list>period_names]
# line search step length bounds
MINIMUM STEP LENGTH = <real>min_step(-infinity)
    [DURING <string list>period_names]
MAXIMUM STEP LENGTH = <real>max_step(infinity)
    [DURING <string list>period_names]
END [CG]

```

Sections 4.2.1 through 4.2.4 describe the components of the CG command block.

4.2.1 Convergence Commands

```
# Default is not defined.
TARGET RESIDUAL = <real>target_resid
    [DURING <string list>period_names]
TARGET RELATIVE RESIDUAL = <real>target_rel_resid(1.0e-4)
    [DURING <string list>period_names]
ACCEPTABLE RESIDUAL = <real>accept_resid(1.0e-3)
    [DURING <string list>period_names]
# Default is 10 times target relative residual
# or 1e20 if a multilevel solver is defined
# and control failure is not defined.
ACCEPTABLE RELATIVE RESIDUAL = <real>accept_rel_resid
    [DURING <string list>period_names]
# Default is not defined.
REFERENCE = EXTERNAL|INTERNAL|BELYTSCHKO|RESIDUAL|ENERGY (EXTERNAL)
    [DURING <string list>period_names]
RESIDUAL NORM TYPE = ALL|TRANSLATION|SCALE_RB_ROTATIONS (ALL)
    [DURING <string list>period_names]
MINIMUM RESIDUAL IMPROVEMENT = <real>resid_improvement
    [DURING <string list>period_names]
MINIMUM ITERATIONS = <integer>min_iter(0)
    [DURING <string list>period_names]
# default = 100 for tangent preconditioner
# default = max(NumNodes,1000) for all other preconditioners.
MAXIMUM ITERATIONS = <integer>max_iter
    [DURING <string list>period_names]
RESIDUAL ROUNDOFF TOLERANCE = <real>1.0e-15)
    [DURING <string list>period_names]
```

The nonlinear preconditioned CG solver iterates to decrease the residual until the residual is deemed to be sufficiently small, based on user-specified convergence criteria. The command lines listed above are placed in the CG command block and used to control these convergence criteria.

Solver convergence is measured by computing the L^2 norm of the residual and comparing that residual norm with target convergence criteria specified by the user. Convergence can be monitored either directly in terms of the residual norm, or in terms of a relative residual, which is the residual norm divided by a reference quantity that is indicative of the current loading conditions on a model. Basing convergence on the relative residual is often helpful because doing so ensures that the convergence target is meaningful for the model. There are some situations, however, when it is better to check for convergence based on the absolute residual rather than on the relative residual. Checking the absolute residual is especially helpful when the model passes through a stage during which there are no loads. The actual residual and the relative residual can be used for checking convergence at the same time.

All the convergence command lines in the `CG` command block can have different values for different time periods by using the `DURING` specification. This specification consists of the key word `DURING` followed by a list of time periods and is appended to the end of the command line. Each of the time periods included in the list must correspond to the name of a `TIME STEPPING` command block (i.e., `time_block_name`) specified in the `TIME CONTROL` command block. If the `DURING` specification is omitted from the command line, the value of the parameter in the command line is applicable for the entire analysis. This value can be overwritten for specific periods by repeating the same command line and using the `DURING` specification. In other words, multiple occurrences of the same command line can be included in the command block, where one without a `DURING` specification applies to many time periods and others with unique `DURING` specifications apply to specific time periods.

The `TARGET RESIDUAL` command line specifies the target convergence criterion in terms of the actual residual norm. The `TARGET RELATIVE RESIDUAL` command line specifies a convergence criterion in terms of the relative residual. If both command lines are included in the `CG` command block, the solver will accept the solution as converged if either the target residual or the target relative residual is below the specified values. Note that use of the term “target” in this discussion means “desired.”

The solver also allows the user to define acceptable convergence criteria. If the solution has not converged to the specified targets before the maximum number of iterations is reached, the residual is checked against the acceptable convergence criteria. These criteria are specified via the `ACCEPTABLE RESIDUAL` and `ACCEPTABLE RELATIVE RESIDUAL` command lines. The concepts of acceptable residual and acceptable relative residual are the same as those used for the target limits, i.e., in the `TARGET RESIDUAL` and `TARGET RELATIVE RESIDUAL` command lines. If the solution has not met the target criteria but has met the acceptable criteria, the solution is allowed to proceed. The default value for each acceptable criterion command line is 10 times the value of its corresponding target criterion command line, e.g., if the value of `target_resid` was $1.0\text{e-}6$, the value of `accept_resid` would be $1.0\text{e-}5$.

When a multilevel solver is being used, the acceptable residual and acceptable relative residual are set to $1\text{e}20$ as long as control failure is not being used. This is because the multilevel controls ultimately check and determine convergence, and allowing the multilevel controls to continue iterating even when a model problem fails to converge can allow the overall problem to converge.

Solutions that meet only the acceptable criteria are noted in the log file. If the acceptable criteria are not met, the solution at that load step has failed to converge, and the solver exits. If adaptive time stepping, as discussed in Section 4.10.2, is active, a solution of the load step may be attempted with a smaller time step. Otherwise, the code will exit at this point with an error.

If relative residuals are given in the convergence criteria, the `REFERENCE` command line can be used to select the method for computing the reference load. This command line has several options: `EXTERNAL`, `INTERNAL`, `BELYTSCHKO`, `RESIDUAL` and `ENERGY`. When the `EXTERNAL` option, which is the default, is selected, the reference load is computed by taking the L^2 norm of the current external load. If the model has force boundary conditions, the external force vector used for this norm is composed of the nodal forces resulting from those boundary conditions. If there are no prescribed forces, the reaction forces at all prescribed kinematic boundary conditions are used instead. The `INTERNAL` option uses the norm of the internal forces as the reference load. This

option is helpful in situations where a structure has no externally applied boundary conditions, such as in the case of a thermally loaded structure. The `BELYTSCHKO` option uses the maximum of the norm of the external load, the norm of the reactions, and the norm of the internal forces. The `RESIDUAL` option denotes that the residual from the initial residual should be used as the reference quantity. The initial residual is computed from the first iteration of the solver.

The `ENERGY` option to the `REFERENCE` command line is a modification of the `EXTERNAL` option where energy conjugate norms of both the residual and reference are used. The energy conjugate norm of the residual is given by $R \cdot K^{-1} \cdot R$ where K is the diagonal of the stiffness matrix. For the energy conjugate norm of the external forces, R is replaced by the external nodal forces. The primary advantage of the energy norm is that forces and moments are converted to consistent and compatible energy units.

Convergence can be difficult in some problems with rigid bodies without any applied moments. In these cases, it has been found advantageous to scale the rotational residuals on the rigid bodies by an approximation of the moment arm of the rigid bodies about the reference point. This can be accomplished through use of the `RESIDUAL NORM TYPE` command with the keyword `SCALE_RB_ROTATIONS`. The other options are `ALL` and `TRANSLATION`. A value of `ALL` is the default and treats all values the same in both the residual norm and the reference norm. A value of `TRANSLATION` uses only the translational components of residual and forces in the norm computations. Though this command is also available for other levels in a multilevel solver, this command is only appropriate for the core CG solver of a Sierra/SM region and should not be used in either RVE or Control Modes regions.

The default `ALL` option means that the residual includes all translational and rotational degrees of freedom. Note, translational forces and rotational moments have different units. Thus the `ALL` option may artificially increase or decrease the contribution of rotational degrees of freedom to the total residual depending on the units used. For example a model with a length unit in millimeters may tend to increase the contribution of rotational degrees of freedom to the convergence check (due to the numerically large moment arm) whereas a model with a length unit of meters may tend to decrease the contribution of rotational degrees of freedom to the convergence check (due to a numerically small moment arm.) The `ALL` option is mathematically inconsistent, but remains the default for historical compatibility. Also in practice the inconsistency is generally irrelevant as translational and rotational degrees of freedom tend to converge at the same rate so that convergence of both quantities is obtained simultaneously.

The `TRANSLATION` option to the `RESIDUAL NORM TYPE` command computes the residual norm over only translational degrees of freedom and ignores all rotational components. This option may be useful if the model is failing to converge rotational degrees of freedom as may occur on problems with very thick shells. The `SCALE_RB_ROTATIONS` option computes the residual over translational degrees of freedom plus scaled rigid body rotational degrees of freedom. The rigid body rotational degrees of freedom are divided by the rigid body characteristic size to produce consistent units for a unified residual norm. The `SCALE_RB_ROTATIONS` option excludes all other rotational degrees of freedom, such as those on shells, from the residual norm. The `SCALE_RB_ROTATIONS` option is recommended if a model contains large rigid bodies with applied external moment boundary conditions. Also note another option to avoid any issues encountered by residual unit incompatibility is to use the `ENERGY` option for the solver `REFERENCE` command.



Warning: The `RESIDUAL NORM TYPE` command should not be used in RVE or Control Modes regions.

The `MINIMUM RESIDUAL IMPROVEMENT` command line stipulates that the CG solver must reduce the initial residual by a specified amount to obtain convergence. If this command line is included in the CG command block, the improvement condition must be met in addition to the standard target criteria. The parameter `resid_improvement` is a real number between 0.0 and 1.0 that specifies the amount by which the residual must be improved as a fraction of the initial residual in the current model problem. Thus, to stipulate that the residual must be smaller than 10% of the initial residual, one would set the value of `resid_improvement` equal to 0.9. The `MINIMUM RESIDUAL IMPROVEMENT` command line is primarily useful in the context of the multilevel solver. The model problems presented to the core solver sometimes begin with low residuals and require few iterations to converge. This command line forces the core solver to further improve the residual, which can accelerate the convergence of the other solver levels.

The `MAXIMUM ITERATIONS` and `MINIMUM ITERATIONS` command lines are used to specify the maximum and minimum number of core nonlinear solver iterations, respectively. These limits apply for a load step if the CG solver is used in the stand-alone mode or for a model problem if the CG solver is used as the core solver in the multilevel solver. The default maximum number of iterations is set to $\max(N_{nodes}, 1000)$ where N_{nodes} is the global number of nodes in the mesh. The linear CG algorithm is guaranteed to converge to the solution in a number of iterations equal to the number of equations in the matrix. However, since we are solving a nonlinear system of equations using a nonlinear CG algorithm, the number of iterations to convergence can be larger than the number of equations. The default value for the minimum number of iterations, `min_iter`, is zero. If a number greater than zero is specified, the solver will iterate at least that many times, regardless of whether the convergence criteria are met.

The `RESIDUAL ROUNDOFF TOLERANCE` command line specifies a tolerance used to compute an approximately zero residual. Due to finite precision arithmetic, the solver can only drop the residual so far before additional corrections to the nodal coordinate field cannot be numerically represented. The approximately zero residual is the residual that would be produced by a perturbation of the displacement by the smallest machine representable perturbation. The default value for this command is the accuracy of double precision arithmetic (1.0e-15). This tolerance number can be set to zero to disable this feature or set higher to converge to larger, but still near machine limit, residuals.

The log file annotates which convergence criterion was met in each CG iteration using markings shown in Table 4.1.

4.2.2 Preconditioner Commands

```
PRECONDITIONER = BLOCK|BLOCK_INITIAL|DIAGONAL|
ELASTIC|IDENTITY|PROBE|TANGENT |
DOMAIN (ELASTIC) [<real>scaling_factor]
PRECONDITIONER ITERATION UPDATE = <integer>iter_update
```

Table 4.1: Log File Convergence Markings

Mark	Convergence Property
>M	Convergence is reached, but still have not iterated the minimum number of specified iterations
>A	Reached maximum iterations and still not converged
<T	Residual converged to target tolerance of either relative or absolute criteria
<A	Reached maximum iterations and converged to acceptable tolerance of either relative or absolute criteria
~0	Residual is converged due to being approximately zero to within machine precision

```

[DURING <string list>period_names]
BALANCE PROBE = <integer>balance_probe(1)
[DURING <string list>period_names]
NODAL PROBE FACTOR = <real>probe_factor(1.0e-6)
[DURING <string list>period_names]
NODAL DIAGONAL SCALE = <real>nodal_diag_scale(0.0)
[DURING <string list>period_names]
NODAL DIAGONAL SHIFT = <real>nodal_diag_shift(0.0)
[DURING <string list>period_names]
BEGIN FULL TANGENT PRECONDITIONER
# ... see Section 4.3
END [FULL TANGENT PRECONDITIONER]

```

These command lines are used to select the type of preconditioner used by the CG algorithm, as well as the method used to form the preconditioner for certain types of preconditioners. The preconditioner is applied to the residual vector to obtain a gradient direction, which in turn is used to find a search direction. The most effective preconditioner is one that closely approximates the effect of multiplying by the inverse of the tangent stiffness matrix.

There are two basic types of preconditioners available: nodal and full tangent. The nodal preconditioners provide only the diagonal terms of the full tangent stiffness matrix or 3-by-3-block diagonal matrices along the diagonal of the stiffness matrix. The diagonal nature of a nodal preconditioner allows it to be inverted inexpensively and also use very little memory. Nodal preconditioners are so termed because they only account for the stiffness at each node in isolation and ignore the coupling between nodes that is accounted for in the off-diagonal terms of a full stiffness matrix. The result of this approximation is that with a nodal preconditioner, in a single CG iteration, the equilibration of a residual at a node can only cause movement in nodes that are directly connected by an element to that node. Thus, as the model size increases, the number of iterations also increases. Nodal preconditioners require many iterations, but they are often efficient because the iterations are inexpensive, especially if the problem is “blocky” in nature.

The `PRECONDITIONER` command line allows the user to select a nodal preconditioner or the full tangent preconditioner with all the defaults. As defined below, this command line has several

options. Some options are synonyms for other options.

- The `BLOCK` and `ELASTIC` options, which are synonyms, specify the default preconditioner. These options provide a nodal preconditioner with 3-by-3-block matrices that are computed based on the elastic material properties. These matrices are updated at every model problem to account for the current geometry.
- The `BLOCK_INITIAL` option, which is a variant of the `BLOCK` preconditioner, forms the preconditioner at the beginning of the analysis but never recomputes it for efficiency.
- The `DIAGONAL` preconditioner is formed in the same way as the `BLOCK` preconditioner, but only uses the terms on the diagonal.
- The `PROBE` option forms a 3-by-3-block diagonal preconditioner by probing the stiffness of nodes rather than by using the elastic stiffness.
- The `IDENTITY` option uses an identity matrix for the preconditioner, meaning that the residual is used as the gradient direction. This option is only of academic interest and is included for completeness.
- The `TANGENT` option is a quick way of specifying the `FULL TANGENT PRECONDITIONER` with all defaults used. See Section 4.3 for a further description of the full tangent preconditioner.
- The `DOMAIN` option is an experimental in development option that should not be used at this time.

The `PRECONDITIONER ITERATION UPDATE` command line controls the frequency at which the full tangent and nodal probe preconditioner are updated. By default, the preconditioner is only updated at the first iteration of each model problem. If this command line is used, updates will occur at the specified frequency `iter_update`. If a model experiences highly nonlinear behavior over the course of a model problem, it may be useful to use this command to cause more frequent preconditioner updates. Such problems may converge better when `iter_update` is set to about 10 for the full tangent case. Realize that frequent updates may reduce the number of CG iterations but increase the computational cost.

The `BALANCE PROBE` and `NODAL PROBE FACTOR` command lines control the behavior of the probing algorithm that is used to obtain the stiffness for the probe preconditioner (selected via the `PROBE` option in the `PRECONDITIONER` command line). For the probe preconditioner, the tangent stiffness K ,

$$K_{ij} = \frac{\partial F_i^{int}}{\partial x_j}, \quad (4.1)$$

where F^{int} is the internal force and x is the displacement, is approximated using finite differences. The nodal probe preconditioner only computes the 3-by-3-block diagonal version of the stiffness matrix. The `NODAL PROBE FACTOR` command line controls the probing distance, with the value of `probe_factor` defaulting to 1.0e-6. The probe factor controls the probing distance relative to element size. Smaller values may give a better tangent approximation, but these values could result in the generation of round-off errors.

The `BALANCE PROBE` command line selects the type of finite differencing scheme used to probe for the stiffness. The value of `balance_probe` can be set to 0, 1, or 2, as explained below.

- Setting `balance_probe` to 0, the default for the nodal probe preconditioner, causes the

preconditioner to be formed by forward finite differencing. Probing occurs in only one direction, resulting in first-order error in the preconditioner. The stiffness computed when `balance_probe` is set to 0 appears as

$$K_{ij} = \frac{F_i^{int}(x + \delta e_j) - F_i^{int}(x)}{\delta}, \quad (4.2)$$

where δ is the probing distance, controlled by the `NODAL PROBE FACTOR` command line, and e_j is a unit vector in the j th equation direction.

- Setting `balance_probe` to 1, the default for the full tangent preconditioner, causes the preconditioner to be formed with central differencing. Probing at each element degree of freedom is performed in both positive and negative directions, leading to an estimate of the tangent stiffness with second-order error. This approach can be necessary in problems with material nonlinearity, where a probe that stretches rather than compresses the element can result in orders of magnitude differences in estimated stiffness. The stiffness computed when `balance_probe` is set to 1 appears as

$$K_{ij} = \frac{F_i^{int}(x + \delta e_j) - F_i^{int}(x - \delta e_j)}{2\delta}. \quad (4.3)$$

- The value of `balance_probe` may also be set to 2, which allows the full tangent preconditioner to obtain central finite differencing with fourth-order error. This approach requires twice the work as central differencing but is more accurate. The stiffness computed when `balance_probe` is set to 2 appears as

$$K_{ij} = \frac{-F_i^{int}(x + 2\delta e_j) + 8F_i^{int}(x + \delta e_j) - 8F_i^{int}(x - \delta e_j) + F_i^{int}(x - 2\delta e_j)}{12\delta}. \quad (4.4)$$

The `NODAL DIAGONAL SCALE` command line is used to specify a nodal diagonal scale factor, which is used to scale the diagonal entries of the nodal preconditioner. This is useful in some situations where a model may be softening due to material effects. The diagonal terms of the nodal preconditioner are scaled by $(1.0 + \text{nodal\_diagonal\_scale})$. The default value of `nodal\_diagonal\_scale` is 0.0, which means the diagonals are not scaled. A value of 0.1 scales up all the diagonal terms by 10 percent.

The `NODAL DIAGONAL SHIFT` command line is used to specify a nodal diagonal shift, which is summed into the diagonal entries of the nodal preconditioner. This is useful in some situations where a model may be softening due to material effects. The diagonal terms of the nodal preconditioner are shifted by $(\text{num\_nodes} * \text{tangent\_diagonal\_shift})$. The default value of `nodal\_diagonal\_shift` is 0.0, which means the diagonals are not shifted.

The `FULL TANGENT PRECONDITIONER` command block enables the full tangent preconditioner and specifies options that apply specifically to that type of preconditioner. See Section 4.3 for a description of the options available. The `FULL TANGENT PRECONDITIONER` command block cannot be used with the `PRECONDITIONER` command line in a `CG` command block. The user must select only one type of preconditioner for an analysis.

4.2.3 Line Search Command

```
LINE SEARCH ACTUAL|TANGENT [DURING <string list>period_names]
```



```

LINE SEARCH SECANT [<real>scale_factor]
LINE SEARCH BACKTRACK [<real>min_step(1.0e-3)
                      <real>reduction(0.95)
                      <real>init_step(1.0)]
LINE SEARCH TYPE MULTI = <real>num_domain(2)
[DURING <string list>period_names]

```

During each CG iteration, after the search direction is computed, a line search is used to find an optimal scaling factor that when applied to the search vector will result in a minimized residual. The line search algorithm is controlled with the `LINE SEARCH` command line. The `SECANT` option is the default, and is used if the `LINE SEARCH` command line is not present.

- The `ACTUAL` line search is a single-step quadratic line search that is equivalent to the corresponding JAS3D line search option. The line search requires an additional evaluation for each iteration of the element internal forces and thus the material response. For a given search direction s , residual r , and velocity v , the step length α is computed as

$$\alpha = -\frac{s^T r(v)}{s^T (F_{int}(v + s) - F_{int}(v))}, \quad (4.5)$$

where the residual is computed based on the internal force F_{int} and on the external force F_{ext} as $r = F_{int}(v) - F_{ext}(v)$.

- The default `SECANT` line search is a slight variation of the `ACTUAL` line search described above. Like the `ACTUAL` line search, this line search requires an additional internal force evaluation for each iteration. The step length α is computed as

$$\alpha = -\frac{s^T r(v)}{s^T (r(v + s) - r(v))}. \quad (4.6)$$

The optional `scale_factor` can be used with the secant line search to scale the search vector s in the equation above. If `scale_factor` is omitted, no scaling is done. If `scale_factor` is provided on the command line, s is scaled by the product of `scale_factor` and the dimension of the smallest element. Setting the scale factor to a small value can be helpful to avoid inverting elements during the evaluation of $r(v + s)$. The `SECANT` line search is recommended for problems in which the external force is highly nonlinear (e.g., a beam-bending problem with a pressure distribution).

- The `TANGENT` line search avoids the additional internal force calculation that is inherent in the other line search types by using an approximate tangent modulus computed in the last call to the material subroutine for each element. The step length α is computed as

$$\alpha = -\frac{s^T r(v)}{s^T K s}, \quad (4.7)$$

where K is the approximate tangent stiffness.

- The `BACKTRACK` line search uses a bisection method to adjust the line search. The solver will first try the `init_step` line search multiplier. If that value is too large the line search multiplier will be scaled by the `reduction` factor and attempted again. This will continue

until either the optimal scaling has been found or this scaling factor reaches the `min_step` value. The `BACKTRACK` line search will tend to be more expensive than other line search methods but has potential more robust in some circumstances such as problems with very sharp nonlinearity.

- The `MULTI` line search first subdivides the line search space into multiple domains and then applies the secant line search method on each domain. The best line search scaling factor found in any sub domain is ultimately used. The multi line search option may in some circumstances provide a more optimal or robust line search value than the secant, though at a significantly increased cost per iteration. The number of subdomains used is defined by the `sub_domain` option which defaults to two.

4.2.4 CG Algorithm Commands

```

ITERATION RESET = <integer>iter_reset(infinity)
    [DURING <string list>period_names]
ORTHOGONALITY MEASURE FOR RESET = <real>ortho_reset(0.5)
    [DURING <string list>period_names]
RESET LIMITS <integer>iter_start <integer>iter_reset
    <real>reset_growth <real>reset_orthogonality
    [DURING <string list>period_names]
BETA METHOD = FletcherReeves|PolakRibiere|Newton|
    PolakRibierePlus(PolakRibiere)
    [DURING <string list>period_names]
MINIMUM STEP LENGTH = <real>min_step(-infinity)
    [DURING <string list>period_names]
MAXIMUM STEP LENGTH = <real>max_step(infinity)
    [DURING <string list>period_names]

```

The behavior of the CG algorithm can be changed using the command lines listed above. The CG algorithm orthogonalizes the current search direction against the previous search direction at each iteration. During an iteration where a reset occurs, the orthogonalization is skipped, resulting in a search in the steepest descent direction. All of these command lines can be appended with the `DURING` specification, as discussed in Section 4.2.1.

The `ITERATION RESET` command line specifies that the CG algorithm be reset after every `iter_reset` iterations. The default value is set equal to the maximum integer, which defaults the conjugate gradient algorithm to not reset.

For the CG algorithm to work well, each new search direction is constructed orthogonal to the previous search directions. If the current search direction has a significant component in one of the previous search directions, error can be introduced that will not be corrected in future iterations, resulting in a lack of convergence. At every iteration, the relative lack of orthogonality between the current gradient direction and the previous search direction is computed. When these become far from orthogonal, a reset can be performed to ignore the previous search directions and use the current gradient direction as the next search direction, which is equivalent to setting the orthogonalization parameter β to zero for that iteration (see Equation 4.8). The `ORTHOGONALITY MEASURE FOR RESET` command line specifies that a reset will occur if the

lack of orthogonality exceeds the value of `ortho_reset`. The default value of 0.5 results in few resets. Setting `ortho_reset` to a value as tight as 0.01 can result in many resets and potentially improved convergence on some problems.

The `RESET LIMITS` command line is also used to control CG resets. This command line is provided to achieve compatibility with JAS3D, and it behaves exactly the same as its JAS3D counterpart. The command line, however, should not be used when either the `ITERATION RESET` command line or the `ORTHOGONALITY MEASURE FOR RESET` command line is included in the CG command block. Up to four parameters can optionally be specified in the `RESET LIMITS` command line. The first, `iter_start`, sets the number of iterations to wait before looking for a minimum residual. The second, `iter_reset`, sets the number of iterations to allow between finding a minimum and restarting the iterative algorithm. The third, `reset_growth`, sets the amount of growth in the residual norm that would indicate divergence and thus trigger a reset. Finally, `reset_orthogonality` sets the relative lack of orthogonality between the current gradient direction and the previous search direction that would trigger a reset.

The `BETA METHOD` command line allows different formulas to be used in computing the orthogonalization parameter β in the CG algorithm,

$$s_k = g_k + \beta s_{k-1}, \quad (4.8)$$

where s_k is the current iteration's search direction, s_{k-1} is the previous iteration's search direction, and g_k is the current iteration's gradient direction.

In the below equations, r_k is the current iteration's residual vector, r_{k-1} is the previous iteration's residual vector, and g_{k-1} is the previous iteration's gradient vector. The `PolakRibierePlus` formula forces beta to be positive. A steepest descent direction is taken when beta becomes negative.

- For the `FletcherReeves` option the β scalar is computed as

$$\beta_k = \frac{r_k^T g_k}{r_{k-1}^T g_{k-1}}. \quad (4.9)$$

- For the `PolakRibiere` option (the default) β is computed as

$$\beta_k = \frac{r_k^T (g_k - g_{k-1})}{r_{k-1}^T g_{k-1}}. \quad (4.10)$$

- For the `PolakRibierePlus` option β is computed as

$$\beta_k = \max(\beta_k^{PR}, 0). \quad (4.11)$$

- For the `Newton` option the nonlinear conjugate gradient algorithm is transformed into a Quasi-Newton method without a line search. In order to best approximate a Newton solve, this method should be used with a full tangent preconditioner that is updated on every iteration.

The `MINIMUM STEP LENGTH` and `MAXIMUM STEP LENGTH` commands set the minimum step length and maximum step length that the CG algorithm can take along a search direction. The default values are set so that any positive or negative step length will be accepted. This option can be used to limit motion of the model along any search direction and can help improve convergence sometimes.

4.2.5 Diagnostic Output Commands

```
ITERATION PRINT = <integer>iter_print  
    [DURING <string list>period_names]  
ITERATION PLOT = <integer>iter_plot  
    [DURING <string list>period_names]  
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks
```

The command lines listed above can be used to control the output of diagnostic information from the CG solver. The `ITERATION PRINT` and `ITERATION PLOT` command lines can be appended with the `DURING` specification, as discussed in Section 4.2.1.

The `ITERATION PRINT` command line controls the frequency at which the convergence information is printed to the log file. The default value of `iter_print` is 25 if a nodal preconditioner is used, and it is 1 if a full tangent preconditioner is used.

For a description of `ITERATION PLOT` and `ITERATION PLOT OUTPUT BLOCKS`, see section 4.1.1.

4.2.6 CG Input Example

```
begin solver  
  Begin cg  
    target relative residual = 1.0e-07  
    Maximum Iterations = 5000  
    preconditioner iteration update = 100  
    preconditioner = elastic  
  end  
end
```

4.3 Full Tangent Preconditioner

In addition to the nodal preconditioners, Sierra/SM provides the capability to use a full tangent preconditioner in the CG algorithm. The full tangent preconditioner employs a scalable parallel linear solver to solve for the gradient direction using the full tangent stiffness matrix. Although the full tangent preconditioner is significantly more costly per CG iteration than the nodal preconditioners, it can solve problems in a smaller number of iterations. Full tangent preconditioners are especially effective in solving poorly conditioned problems that are often extremely difficult to solve using the CG algorithm with nodal preconditioners, such as those problems that involve the bending response of long, slender members.

To use a full tangent preconditioner generally two command blocks are added to the input file, one required and one optional. First, the `FULL TANGENT PRECONDITIONER` command block, described in this section, must be added to the CG command block. The `FULL TANGENT PRECONDITIONER` command block is used instead of the `PRECONDITIONER` command line in the CG command block. Second, a command block for a linear solver may be defined in the SIERRA scope, as described in Section 4.4. That linear solver command block may then be referenced from within the `FULL TANGENT PRECONDITIONER` command block, instructing the CG algorithm to use the specified linear solver. If no linear solver command block is given and/or no linear solver is referenced in the `FULL TANGENT PRECONDITIONER` command block, the FETI linear solver will be employed with default settings. The command block to enable the full tangent preconditioner is as follows:



Known Issue: Deactivation of element blocks (see Section 6.1.6.9) in conjunction with the full tangent preconditioner is available but not yet fully tested. Use caution when combining these capabilities.

```
BEGIN FULL TANGENT PRECONDITIONER
#
# solver selection commands
LINEAR SOLVER = <string>linear_solver_name(FETI)
[DURING <string list>period_names]
NODAL PRECONDITIONER METHOD = ELASTIC|PROBE|DIAGONAL|BLOCK
(ELASTIC)
#
# tangent matrix formation commands
PROBE FACTOR = <real>probe_factor(1.0e-6)
[DURING <string list>period_names]
BALANCE PROBE = <integer>balance_probe(1)
[DURING <string list>period_names]
CONSTRAINT ENFORCEMENT = SOLVER|PENALTY(PENALTY)
PENALTY FACTOR = <real>penalty_factor(100.0)
[DURING <string list>period_names]
# useful to improve ill-conditioned tangent matrix
TANGENT DIAGONAL SCALE = <real>tangent_diag_scale(0.0)
```

```

    [DURING <string list>period_names]
TANGENT DIAGONAL SHIFT = <real>tangent_diag_shift(0.0)
    [DURING <string list>period_names]
# turn off for speed
CONDITIONING = NO_CHECK|CHECK|AUTO_REGULARIZATION(CHECK)
    [DURING <string list>period_names]
HANGING NODE FACTOR = <real>(1.0e-3)
    [DURING <string list>period_names]
#
# reset and iteration commands
MAXIMUM RESETS FOR MODELPROBLEM = <integer>max_mp_resets
    (100000) [DURING <string list>period_names]
MAXIMUM RESETS FOR LOADSTEP = <integer>max_ls_resets
    (100000) [DURING <string list>period_names]
MAXIMUM ITERATIONS FOR MODELPROBLEM
    = <integer>max_mp_iter(100000)
    [DURING <string list>period_names]
MAXIMUM ITERATIONS FOR LOADSTEP = <integer>max_ls_iter
    (100000) [DURING <string list>period_names]
ITERATION UPDATE = <integer>iter_update
    [DURING <string list>period_names]
# freeze current preconditioner
SMALL NUMBER OF ITERATIONS = <integer>small_num_iter
    [DURING <string list>period_names]
MINIMUM SMOOTHING ITERATIONS = <integer>min_smooth_iter(0)
    [DURING <string list>period_names]
MAXIMUM SMOOTHING ITERATIONS = <integer>max_smooth_iter(0)
    [DURING <string list>period_names]
TARGET SMOOTHING RESIDUAL = <integer>tgt_smooth_resid(0)
    [DURING <string list>period_names]
TARGET SMOOTHING RELATIVE RESIDUAL
    = <integer>tgt_smooth_rel_resid(0)
    [DURING <string list>period_names]
#
# fall-back strategy commands
STAGNATION THRESHOLD = <real>stagnation(1.0e-12)
    [DURING <string list>period_names]
# triggers preconditioner switch or update
MINIMUM CONVERGENCE RATE = <real>min_conv_rate(1.0e-4)
    [DURING <string list>period_names]
# either switch to nodal preconditioner or update
ADAPTIVE STRATEGY = SWITCH|UPDATE(SWITCH)
    [DURING <string list>period_names]

END [FULL TANGENT PRECONDITIONER]

```

The FULL TANGENT PRECONDITIONER command block contains all command lines related to

the interaction with the linear solver. There are command lines to control selection of the linear solver, formation of the matrices passed into the linear solver, CG iteration strategies, updating strategies, and strategies for dealing with poor convergence. Reasonable defaults have been set for most problems. The command lines in this command block are described in Sections [4.3.1](#) through [4.3.5](#).

4.3.1 Solver Selection Commands

```

LINEAR SOLVER = <string>linear_solver_name
                [DURING <string list>period_names]
NODAL PRECONDITIONER METHOD = ELASTIC|PROBE|DIAGONAL|
                              BLOCK(ELASTIC)

```

The command lines listed above are used in selecting a linear solver for use with the full tangent preconditioner and in selecting a nodal preconditioner. The `LINEAR SOLVER` command line specifies the name of the linear solver used to compute the action of the preconditioner by solving the linear system. If no linear solver is specified, Sierra/SM will default to using the FETI linear solver with its default settings. Linear solvers are defined in the SIERRA scope.

Although several linear-solver packages are available for use as preconditioners in Sierra/SM, we recommend that the FETI equation solver be used in production analyses. The FETI equation solver is actively maintained and tested by the Sierra/SM development team. The `FETI EQUATION SOLVER` command block is documented in Section [4.4](#). Other linear solvers may have compatibility problems with other code capabilities such as MPCs and contact.

An arbitrary number of equation solver command blocks can be included in the input file to define various sets of solver options. Each equation solver command block must be given a unique name, which is referenced in the `linear_solver_name` parameter of the `LINEAR SOLVER` command line. This name selects the specified linear solver as a preconditioner for the CG solver.

When Sierra/SM uses a full tangent preconditioner, it also forms a nodal preconditioner. This nodal preconditioner is used in contact calculations, as a fall-back preconditioner if the full tangent preconditioner does not perform well and optionally for performing smoothing iterations before use of the full tangent preconditioner.

The nodal preconditioner is selected with the `NODAL PRECONDITIONER METHOD` command line. Four options are available: `ELASTIC`, `PROBE`, `DIAGONAL`, and `BLOCK`. The `ELASTIC` option is the default. These options have the same meaning as the corresponding options given for the nodal preconditioner in the `CG` command block, as documented in Section [4.2.2](#).

4.3.2 Matrix Formation Commands

```

PROBE FACTOR = <real>probe_factor(1.0e-6)
                [DURING <string list>period_names]
BALANCE PROBE = <integer>balance_probe(1)
                [DURING <string list>period_names]
CONSTRAINT ENFORCEMENT = SOLVER|PENALTY(PENALTY)
PENALTY FACTOR = <real>penalty_factor(100.0)
                [DURING <string list>period_names]

```

```

TANGENT DIAGONAL SCALE = <real>tangent_diag_scale(0.0)
[DURING <string list>period_names]
TANGENT DIAGONAL SHIFT = <real>tangent_diag_shift(0.0)
[DURING <string list>period_names]
CONDITIONING = NO_CHECK|CHECK|AUTO_REGULARIZATION(CHECK)
[DURING <string list>period_names]
HANGING NODE FACTOR = <real>(1.0e-3)
[DURING <string list>period_names]

```

The command lines listed above can be used to optionally control the way the stiffness matrix is formed. This matrix is formed by assembling contributions from all active elements. These element stiffness matrices are formed by finite differencing.

The `PROBE FACTOR` command line controls the finite difference probing distance relative to element size. The default value of `probe_factor` is 1.0e-6. Smaller values may give a better tangent approximation, but they may also generate round-off errors.

The `BALANCE PROBE` command line selects the type of finite-differencing scheme used to probe for the stiffness. The value of `balance_probe` can be set to 0, 1, or 2. Setting `balance_probe` to 0 causes the preconditioner to be formed by forward finite differencing. Probing occurs in only one direction, resulting in first-order truncation error in the preconditioner. Setting `balance_probe` to 1, the default for the full tangent preconditioner, causes the preconditioner to be formed with central differencing. Probing at each element degree of freedom is performed in both positive and negative directions, leading to a second-order estimate of the tangent stiffness. The value of `balance_probe` may also be set to 2, which allows the full tangent preconditioner to obtain central finite differencing with fourth-order error. This scheme requires twice the work as the central-differencing scheme but is more accurate. See Section 4.2.2 for a more detailed discussion of these options.

The `CONSTRAINT ENFORCEMENT` command line controls the way multipoint constraints (i.e., contact, rigid body, and periodic boundary conditions) are enforced in the linear system. Available options are `SOLVER` and `PENALTY`. When `SOLVER` is specified, constraints are passed to the solver, and constraint enforcement is handled internally. The `PENALTY` specification converts constraints to penalty “elements” whose stiffnesses (controlled by the `PENALTY FACTOR` command line—see below) are contributed to the linear system directly.

Although some of the available linear-solver packages support internal constraint enforcement, these packages often perform poorly in parallel. Thus, it is recommended that `CONSTRAINT ENFORCEMENT` be set to `PENALTY` for all analyses. The errors caused by penalty compliance are corrected in each CG iteration, so the converged solution will not have errors due to penalty compliance.

The `PENALTY FACTOR` command line is used to specify the penalty stiffness. At each constraint, the diagonal tangent stiffness of the slave degree of freedom is multiplied by the value of the `penalty_factor` parameter. The default value for this parameter is 100.0, which is reasonable for most problems. Assigning a high value (greater than about 1.0e+8) can cause poor conditioning of the linear system and also cause the linear solver to perform poorly. Assigning a low value (less

than about 10) could require more CG iterations due to the correction of errors resulting from penalty compliance.

The `TANGENT DIAGONAL SCALE` command line is used to specify a tangent matrix diagonal scale factor, which is used to scale the diagonal entries of the tangent matrix. This is useful in some situations where a model does not have sufficient boundary conditions to remove all rigid body modes. This command has also been used as a last resort strategy to achieve convergence when all else fails. The diagonal terms of the tangent matrix are scaled by $(1.0 + \text{tangent\_diagonal\_scale})$. The default value of `tangent\_diagonal\_scale` is 0.0, which means the tangent matrix diagonals are not scaled. A value of 0.1 scales up all tangent matrix diagonal terms by 10 percent.

The `TANGENT DIAGONAL SHIFT` command line is used to specify a tangent matrix diagonal shift, which is summed into the diagonal entries of the tangent matrix. This is useful in some situations where a model does not have sufficient boundary conditions to remove all rigid body modes. This command has also been used as a last resort strategy to achieve convergence when all else fails. The diagonal terms of the tangent matrix are shifted by $(\text{num\_nodes} * \text{tangent\_diagonal\_shift})$. The default value of `tangent\_diagonal\_shift` is 0.0, which means the tangent matrix diagonals are not shifted.

The `CONDITIONING` command line controls the linear system conditioning inside the FETI solver. It has no effect on any other linear system solver. By default, when used with the full tangent preconditioner FETI checks the conditioning of its coarse level matrix and reports a warning to the log file if it finds ill-conditioning (defined as a condition number greater than $1e5$, where the condition number is defined as the ratio of the maximum eigenvalue to the minimum eigenvalue). This default behavior corresponds to the `CHECK` option. For performance optimization this check may be turned off using the `NO_CHECK` option. The third option, `AUTO_REGULARIZATION`, instructs FETI to add a small shift to its coarse matrix to regularize the linear system and artificially reduce the condition number. In the context of the CG solver, this will affect convergence but not the residual calculations. Coarse level matrix ill-conditioning often indicates an issue with the problem, such as the presence of zero energy rigid body modes either inherently or temporarily as a result of a CG iteration, and `AUTO_REGULARIZATION` may improve CG convergence in the presence of ill-conditioning. A basic power method is used to compute the highest eigenvalue, and a shifted power method is used to compute the lowest eigenvalue. This capability should be viewed both as a diagnostic tool and a robustness aide.

The `HANGING NODE FACTOR` command is used to apply a nodal stiffness to any nodes that may exist on a processor but are not attached to any element on that processor. This configuration can occur with parallel contact, spot welds, MPCs, and other non-local joints and boundary conditions. This artificial stiffness is used to avoid singularities in the processor local stiffness matrix. The value used is a scale factor on the diagonal nodal stiffness.

4.3.3 Reset and Iteration Commands

```
MAXIMUM RESETS FOR MODELPROBLEM = <integer>max_mp_resets
    (100000) [DURING <string list>period_names]
MAXIMUM RESETS FOR LOADSTEP = <integer>max_ls_resets
    (100000) [DURING <string list>period_names]
MAXIMUM ITERATIONS FOR MODELPROBLEM = <integer>max_mp_iter
```

```

(100000) [DURING <string list>period_names]
MAXIMUM ITERATIONS FOR LOADSTEP = <integer>max_ls_iter
(100000) [DURING <string list>period_names]
ITERATION UPDATE = <integer>iter_update
[DURING <string list>period_names]
SMALL NUMBER OF ITERATIONS = <integer>small_num_iter(0)
[DURING <string list>period_names]

```

When a linear solver is used as a preconditioner, the efficiency of the solution can often be dramatically affected by the frequency at which the preconditioner is updated. The computational expense of updating the preconditioner, which involves forming and factorizing the stiffness matrix, is often much higher than the cost of applying the preconditioner, which involves only a back-solve, during a CG iteration. If the stiffness does not dramatically change from iteration to iteration, it is often much more efficient to re-use the preconditioner for many iterations, or even load steps, than it is to update it at every iteration.

The command lines listed above can be used to control when the preconditioner is updated. As Sierra/SM allows for the nodal preconditioner to be used in place of the full tangent preconditioner in some situations, these command lines also control this behavior. All of these command lines can be appended with the `DURING` specification, as discussed in Section 4.2.1.

The `MAXIMUM RESETS FOR MODELPROBLEM` command line limits the number of times that the preconditioner can be reset (updated) during a model problem. By default, there is no limit on the number of updates that can occur during a model problem. The `MAXIMUM RESETS FOR LOADSTEP` command line limits the updates that can occur during a load step. By default, there is also no limit on the number of updates that can occur during a load step.

The `MAXIMUM ITERATIONS FOR LOADSTEP` and `MAXIMUM ITERATIONS FOR MODELPROBLEM` command lines can be used to limit the number of CG iterations performed using the full tangent preconditioner per model problem or load step, respectively. The values of the parameters `max_mp_iter` and `max_ls_iter` are both unlimited by default. If one of these iteration limits is reached, the CG solver will continue to iterate using the fall-back nodal preconditioner until either the CG solver iteration limit is reached or convergence is achieved. It can be useful to limit the iterations taken by the full tangent preconditioner. If the full tangent preconditioner is performing poorly, the nodal preconditioner may work better sometimes.

The `ITERATION UPDATE` command line controls the frequency at which the full tangent preconditioner is updated. By default, it is only updated at the first iteration of each model problem. If this command line is used, the preconditioner will be updated at the frequency specified by `iter_update`. If a model experiences highly nonlinear behavior over the course of a model problem, it may be useful to use this command line to cause more frequent preconditioner updates. Such problems may converge better when `iter_update` is set to about 10. Realize that frequent updates may reduce the number of CG iterations but dramatically increase the computational cost.

It can often be efficient to update the preconditioner infrequently. The `SMALL NUMBER OF ITERATIONS` command line is used to “freeze” the preconditioner, or prevent updates, until a model problem requires more iterations than the number specified in `small_num_iter`. The default value of `small_num_iter` is 0, meaning that the preconditioner is never re-used for the next

model problem unless the `SMALL NUMBER OF ITERATIONS` command line is used. It is often beneficial to set `small_num_iter` to 10. This can result in the re-use of the preconditioner over many load steps. When this command line is used, the CG solver may converge in a very small number of iterations immediately following a preconditioner update; however, over the course of several load steps, the iteration counts slowly increase because the preconditioner is out of date. At a certain point, the iteration count exceeds `small_num_iter`, and the preconditioner is reset. This solution strategy is efficient because it allows re-use of the preconditioner while it is still relatively effective.



Warning: The `SMALL NUMBER OF ITERATIONS` command is not guaranteed to be compatible with linear solvers other than FETI. For instance, the Aztec solver fails to find the underlying preconditioner when it is used in an aria and an adagio region. A possible workaround is to use a unique linear solver in each region.

4.3.4 Smoothing Iterations

```

MINIMUM SMOOTHING ITERATIONS = <integer>min_smooth_iter(0)
[DURING <string list>period_names]
MAXIMUM SMOOTHING ITERATIONS = <integer>max_smooth_iter(0)
[DURING <string list>period_names]
TARGET SMOOTHING RESIDUAL = <integer>tgt_smooth_resid(0)
[DURING <string list>period_names]
TARGET SMOOTHING RELATIVE RESIDUAL =
  <integer>tgt_smooth_rel_resid(0)
[DURING <string list>period_names]
AUTOMATIC SMOOTHING FACTOR = <real>auto_smooth
AUTOMATIC SMOOTHING RELATIVE RESIDUAL FACTOR = <real>rel_auto_smooth
AUTOMATIC SMOOTHING RESIDUAL FACTOR = <real>abs_auto_smooth

```

Sometimes it is beneficial to perform many CG iterations using the nodal preconditioner before applying the full tangent preconditioner. The iterations with the nodal preconditioner are used as a “smoother” to put the model in a better state for the full tangent preconditioner. The smoothing iterations are controlled using the `MINIMUM SMOOTHING ITERATIONS`, `MAXIMUM SMOOTHING ITERATIONS`, `TARGET SMOOTHING RESIDUAL`, and `TARGET SMOOTHING RELATIVE RESIDUAL` command lines. By default, no smoothing iterations are taken. Using these commands, the user can specify that smoothing iterations should be taken and control how many are taken.

The number of smoothing iterations taken can vary based on the convergence of the solver during those iterations. The smoothing iteration commands allow the user to control the desired convergence of the smoother before switching to the full tangent preconditioner. These commands behave the same way as the standard solver convergence commands, except that they apply only to the smoothing iterations. The standard convergence criteria still must be met to obtain a converged solution. The `MINIMUM SMOOTHING ITERATIONS` command controls the minimum smoothing iterations taken, and the `MAXIMUM SMOOTHING ITERATIONS` com-

mand controls the maximum smoothing iterations. The `TARGET SMOOTHING RESIDUAL` and `TARGET SMOOTHING RELATIVE RESIDUAL` control the residual and relative residual, respectively, that must be obtained in the smoothing iterations before switching to the full tangent preconditioner. Once `tgt_smooth_resid` has been reached, the solver switches from using the nodal preconditioner to the tangent preconditioner. If `max_smooth_iter` is reached before the target smoothing convergence is reached, the solver simply switches to using the full tangent preconditioner.

This feature can also be used in cases where the nodal preconditioner may be effective enough to achieve convergence with relatively few iterations, but occasionally the nodal preconditioner is not effective. If `max_smooth_iter` is set to a relatively high number, many load steps can be solved without ever using the full tangent preconditioner, but this preconditioner is used on the more difficult steps that require more iterations.

Another use of the smoothing iteration control commands is for switching between the nodal preconditioner and the full tangent preconditioner for certain time periods. These command lines, like all the other command lines discussed above, can be set on a period-specific basis by appending them with the `DURING` keyword, followed by a list of periods. To use the nodal preconditioner on some periods, the user can set `max_smooth_iter` to a high number for those periods and leave the default value of 0 for the periods in which the full tangent preconditioner is desired.

An alternative way the smoothing tolerances can be set is with the `AUTOMATIC SMOOTHING FACTOR` commands. The automatic smoothing factor commands automatically select an appropriate smoothing residual target based on the initial and target residuals.

The `AUTOMATIC SMOOTHING RELATIVE RESIDUAL FACTOR` effectively sets the `TARGET SMOOTHING RELATIVE RESIDUAL` r_S based off of the initial relative residual r_0 , target relative residual r_T and input `rel_auto_smooth` scaling factor f using the following equation.

$$r_S = e^{f*(\ln(r_T)-\ln(r_0))+\ln(r_0)} \quad (4.12)$$

Similarly The `AUTOMATIC SMOOTHING RESIDUAL FACTOR` effectively sets the `TARGET SMOOTHING RESIDUAL` R_S based off of the initial residual R_0 , target residual R_T and input `abs_auto_smooth` scaling factor F using the following equation.

$$R_S = e^{F*(\ln(R_T)-\ln(R_0))+\ln(R_0)} \quad (4.13)$$

Use of the `AUTOMATIC SMOOTHING FACTOR` command sets both the `AUTOMATIC SMOOTHING RELATIVE RESIDUAL FACTOR` and the `AUTOMATIC SMOOTHING RESIDUAL FACTOR` at the same time to the same value.

When using the `AUTOMATIC SMOOTHING FACTOR` command the scaling factor may be any number between zero and one though appropriate values are generally in the range of 0.1 to 0.5.

4.3.5 Fall-Back Strategy Commands

```
STAGNATION THRESHOLD = <real>stagnation(1.0e-12)
MINIMUM CONVERGENCE RATE = <real>min_conv_rate(1.0e-4)
ADAPTIVE STRATEGY = SWITCH|UPDATE(SWITCH)
```

Occasionally, the full tangent preconditioner may stagnate, failing to significantly reduce the residual from one CG iteration to the next. At each iteration, Sierra/SM checks for slow convergence, and attempts to remedy poor convergence if detected. The commands listed above can optionally be added to the `FULL TANGENT PRECONDITIONER` command block.

The `STAGNATION THRESHOLD` and `MINIMUM CONVERGENCE RATE` command lines are used to set the `stagnation` and `min_conv_rate` parameters, respectively. These parameters are used in a strategy that attempts to remedy slow convergence. The method for remedying slow convergence is controlled with the `ADAPTIVE STRATEGY` command line, which can have a value of `SWITCH` or `UPDATE`. The default strategy is to `SWITCH` from the tangent preconditioner to the nodal preconditioner. The formula for computing the convergence rate is given by

$$abs((\|R_{n-1}\| - \|R_n\|)/\|R_{n-1}\|) \quad (4.14)$$

where R is the residual force and n is the iteration counter. In the `SWITCH` strategy, if the convergence rate is less than the value of `min_conv_rate` but greater than the value of `stagnation`, the CG solver will switch to using the fall-back nodal preconditioner. In the `UPDATE` strategy, the full tangent preconditioner would be updated instead. In either case, if the convergence rate is below the value of `stagnation`, the solver will switch to the nodal preconditioner.

4.3.6 CG with Full Tangent Preconditioner Input Example

```
begin solver
  Begin cg
    target relative residual = 1.0e-06
    Maximum Iterations      = 100
    # uses feti as the default linear equation solver
    begin full tangent preconditioner
      tangent diagonal scale = 1e-6
    end
  end
end
```

4.4 FETI Equation Solver

FETI is a domain-decomposition-based parallel iterative linear solver that can be used to compute the action of the global tangent stiffness matrix for the nonlinear CG solver [3, 4]. FETI uses a direct solver on each domain and iteratively solves for Lagrange multiplier fields at the domain boundaries.

FETI also generates a coarse grid matrix from the global tangent stiffness matrix. The coarse matrix is used to accelerate convergence and distribute the error more evenly across the local mesh domains. The highest eigenvalue λ_1 and lowest eigenvalue λ_N of the coarse matrix are relatively inexpensive to calculate. These eigenvalues are then used to compute the condition number of the coarse matrix $\kappa = \lambda_1/\lambda_N$. If κ is excessively large, then a warning is printed to the log file, indicating that the problem is ill-posed.

Under typical usage, the FETI domains correspond to the portions of the model owned by each processor. If a model is run on a single processor, FETI simply behaves as a direct solver. Because large models are typically run using many processors, FETI uses significantly lower total computing resources than a direct solution method of the whole problem would require. The large number of small direct solutions may be performed in parallel.

Although a number of other linear solvers are available for use with the full tangent preconditioner in the CG solver, it is recommended that FETI be used. FETI is actively maintained and tested by the Sierra/SM development team. FETI has been demonstrated as a robust parallel solver has been used successfully on a wide range of production analyses. The command block for the FETI equation solver is as follows:

```
BEGIN FETI EQUATION SOLVER <string>name
#
# convergence commands
MAXIMUM ITERATIONS = <integer>max_iter(500)
RESIDUAL NORM TOLERANCE = <real>residual_tol(1.0e-3)
#
# diagnostic commands, off by default.
# "solver" turns on print statements from FETI
# "matrix" dumps the matrix to a MATLAB file (in serial)
PARAM-STRING "debugMask" VALUE <string>"solver"|"matrix"
#
# Hinge detection (bad decomposition topologies)
PARAM-STRING "detect_bad_topology" VALUE <string>"true"
#
# Dumps cubit journal files with bad topologies for viz
PARAM-STRING "debug_bad_topology" VALUE <string>"true"
#
# memory usage commands
PARAM-STRING "precision" VALUE <string>"single"|"double"
    ("double")
#
# lumped is uses less memory, Dirichlet improves convergence
```

```

PRECONDITIONING METHOD = NONE|LUMPED|DIRICHLET(DIRICHLET)
MAXIMUM ORTHOGONALIZATION = <integer>max_orthog(1000)
#
# solver commands
LOCAL SOLVER = SKYLINE|SPARSE|ITERATIVE(SPARSE)
COARSE SOLVER = SKYLINE|SPARSE|ITERATIVE(SPARSE)
#
# This command is only used in conjunction with
# local solver = iterative.
NUM LOCAL SUBDOMAINS = <integer>num_local_subdomains
#
# factorization tolerance for local domain matrix
PARAM-REAL "local_rbm_tol" value <real>(1e-12)
#
# factorization tolerance for global coarse matrix
PARAM-REAL "global_rbm_tol" value <real>(1e-9)

END [FETI EQUATION SOLVER <string>name]

```

The command lines used to control FETI all reside in the `FETI EQUATION SOLVER` command block, where `name` identifies the particular command block. This command block, as with all equation solver command blocks, is placed in the SIERRA scope and referenced by `name` when it is used. Although a number of command lines are available to control the behavior of FETI, the default settings generally work well for the vast majority of problems. Thus, it is recommended that all default settings be used unless special behavior is desired because of the unique features of a specific model. The command lines in the `FETI EQUATION SOLVER` command block are described in Sections [4.4.1](#) through [4.4.3](#).

4.4.1 Convergence Commands

```

MAXIMUM ITERATIONS = <integer>max_iter(500)
RESIDUAL NORM TOLERANCE = <real>residual_tol(1.0e-3)

```

These command lines provide controls on the convergence of the FETI iterative solver and belong in the `FETI EQUATION SOLVER` command block.

The `MAXIMUM ITERATIONS` command line sets the maximum number of iterations allowed per FETI solve. The default value of the parameter `max_iter` is 500. A FETI solve occurs once per CG iteration when used with the full tangent preconditioner.

The `RESIDUAL NORM TOLERANCE` command line sets the convergence criterion for the FETI solver. The default value of the parameter `residual_tol` is 1.0e-3. The relative residual norm is compared against the linear residual $\|Ax - b\|$ divided by the norm of current residual, $\|b\|$ where b is the residual force, A is the tangent stiffness matrix, and x is the linear solution vector. If convergence is not reached before the iteration count exceeds `max_iter`, FETI will simply return the current approximate gradient direction to the CG solver and continue iterating. The code will not exit with an error. The default settings for both of these command lines are reasonable for most models and typically should not be modified.

4.4.2 Memory Usage Commands

```
PARAM-STRING "precision" VALUE <string>"single"|"double"  
("double")  
PRECONDITIONING METHOD = NONE|LUMPED|DIRICHLET(DIRICHLET)  
MAXIMUM ORTHOGONALIZATION = <integer>max_orthog(1000)
```

The command lines listed above can be placed in the `FETI EQUATION SOLVER` command block to enable optional memory-saving features of the FETI solver. All of these features will adversely impact the solver effectiveness to some degree, but they can be useful if the memory requirements of a model exceed the capacity of the machine on which the model is run. Before using these features, it is important to consider that on a distributed memory cluster, spreading the model out over more processors can reduce the memory requirements on each processor. For this reason, it is often better to use more processors rather than use the options described here.

FETI has the option of using either single or double precision for storage of internal variables. The default behavior is to use double precision, and this is typically recommended. To select single precision, the user would specify the command line as follows: `PARAM-STRING "precision" VALUE "single"`. Using single-precision variables within FETI can reduce the memory requirements by roughly a factor of two. Use of single precision may, however, degrade the performance of the solver, requiring more iterations within FETI or more CG iterations. Using single precision in FETI does not affect the Sierra/SM data structures, which are always double precision, and therefore does not adversely affect solution ultimate accuracy.

The `PRECONDITIONING METHOD` command line selects the preconditioning method that is used internally within FETI. The default option, `DIRICHLET`, typically results in the best convergence rate. The `LUMPED` option uses less memory, but it usually results in more iterations within FETI. This option should only be used if there are constraints on memory usage. The `NONE` option uses no preconditioner and is included only for completeness. This option is not of practical interest.

Like the CG solver, the FETI equation solver stores a set of search directions to ensure that the search direction used in each iteration is orthogonal to previous search directions. The number of search directions stored is controlled with the `MAXIMUM ORTHOGONALIZATION` command line. The default value of 1000 for `max_orthog` provides optimal convergence for linear solves that take less than 1000 iterations. Setting `max_orthog` to a lower number can decrease memory usage however, as with the other options discussed above, may require more iterations for convergence.

4.4.3 Solver Commands

```
LOCAL SOLVER = SKYLINE|SPARSE|ITERATIVE(SPARSE)  
COARSE SOLVER = SKYLINE|SPARSE|ITERATIVE(SPARSE)  
NUM LOCAL SUBDOMAINS = <integer>num_local_subdomains  
PARAM-REAL "local_rbm_tol" value <real>(1e-12)  
PARAM-REAL "global_rbm_tol" value <real>(1e-9)
```

These command lines control the type of solver used by FETI for solving the linear system that arises from the coarse grid and for the local solution mesh linear system. The `LOCAL SOLVER` command line is used to select the solver for the local subdomains. Similarly, the solver for the

coarse grid can be selected with the `COARSE SOLVER` command lines. Both of these command lines allow the same three options: `SPARSE`, `SKYLINE`, and `ITERATIVE`, where `SPARSE` is the typical default. The coarse solver default is set based on the number of processors being used in the problem. For processor counts less than 3, the `SKYLINE` solver is the default coarse grid solver. For processor counts greater than 3 and less than 2000, the `SPARSE` solver is the default coarse grid solver. For processor counts greater than 2000, the `ITERATIVE` coarse grid solver is the default.

The FETI algorithm uses these direct solvers to factorize the local matrix and coarse grid matrices before iterating on the solution. These factorization algorithms use tolerances to detect singularities in the matrix. The default values for `local_rbm_tol` and `global_rbm_tol` are recommended in most situations.

- The `SPARSE` option uses a sparse matrix storage direct solver. The sparse matrix is factored into an $A = LDL^T$ decomposition. FETI takes advantage of equation orderings to reduce matrix fill-in. The default equation ordering is done by calling the METIS sparse matrix ordering algorithm. This solver is recommended for both speed and memory efficiency.
- The `SKYLINE` option uses a skyline (profile) matrix storage direct solver. The skyline matrix is factored into an $A = LDL^T$ decomposition. This method is robust, detects rigid-body modes effectively, and includes the reverse Cuthill-McKee (RCM) and Sloan equation orderings. This solver uses more memory than does the sparse solver.
- The `ITERATIVE` option implements a multiple domain FETI algorithm for the local solver or the coarse solver. The number of domains for the local solver is either computed based on internal heuristics or input by the user. The number of domains for the coarse solver is defaulted to 2 which means that the iterative coarse solver will run on 2 processors. This option is recommended in special circumstances where memory becomes an issue with the sparse direct solver or the problem is being run on greater than 2000 processors.

The `NUM LOCAL SUBDOMAINS` command line sets the number of local subdomains for the `ITERATIVE` local solver. This command line is not used for the other local solvers. As the number of local subdomains increases, the size of the local subdomains decreases, thus reducing the memory requirements and the time it takes for the local matrix factorizations. As the number of subdomains increases, the size of the coarse grid increases, thus requiring increased factorization time and memory. A good rule of thumb is to set the number of local subdomains to the total number of elements in the mesh divided by 400. For a serial run, if the `NUM LOCAL SUBDOMAINS` command line is not specified, FETI sets the number of domains to the greater of 2 or the number of elements divided by 400. For parallel runs, FETI uses the same number of domains as that used by Sierra/SM.

4.4.4 CG with Full Tangent Preconditioner and FETI Input Example

The following example defines a feti solver for use in a CG preconditioner. The linear solver `my_feti` references a feti solver with a non-default parameter set. This example sets the preconditioning method to the lumped method to reduce memory usage.

```
Begin solver
  Begin cg
    target relative residual = 1.0e-09
```

```

        Maximum Iterations      = 100
        begin full tangent preconditioner
            linear solver = my_feti
        end
    end
end
end adagio region adagio
end adagio procedure Agio_Procedure

begin feti equation solver my_feti
    preconditioning method = lumped
end

```

4.5 Control Contact

The multilevel solution control scheme used for contact is called control contact. Control contact alleviates some of the challenges associated with the nonlinear effects of contact by incrementally enforcing the contact nonlinearity outside the core solver. The specific steps taken during a control contact iteration depends on whether augmented Lagrange or kinematic enforcement is being used. Generally, the interactions (node-face or face-face pairs) that are being enforced are only changed once at the start of each control contact iteration and then held fixed during the core CG solve.

The command block for control contact is:

```
BEGIN CONTROL CONTACT
#
# Convergence commands
TARGET RESIDUAL = <real>target_resid
    [DURING <string list>period_names]
TARGET RELATIVE RESIDUAL = <real>target_rel_resid(1.0e-3)
    [DURING <string list>period_names]
TARGET RELATIVE CONTACT RESIDUAL =
    <real>target_rel_cont_resid
    [DURING <string list>period_names]
ACCEPTABLE RESIDUAL = <real>accept_resid
    [DURING <string list>period_names]
ACCEPTABLE RELATIVE RESIDUAL = <real>accept_rel_resid(1.0e-2)
    [DURING <string list>period_names]
ACCEPTABLE RELATIVE CONTACT RESIDUAL =
    <real>accept_rel_cont_resid(1.0e-1)
    [DURING <string list>period_names]
MINIMUM ITERATIONS = <integer>min_iter(0)
    [DURING <string list>period_names]
MAXIMUM ITERATIONS = <integer>max_iter(100)
    [DURING <string list>period_names]
#
# level selection command
LEVEL = <integer>control_contact_level(1)
#
# Augmented Lagrange enforcement commands used with
# enforcement = al defined in the contact definition
#
# Adaptive Penalty Options
#
LAGRANGE INITIALIZE = NONE|MULTIPLIER|PENALTY|BOTH (MULTIPLIER)
    [DURING <string list>period_names]
LAGRANGE ADAPTIVE PENALTY = OFF|SEPARATE|UNIFORM (UNIFORM)
    [DURING <string list>period_names]
LAGRANGE ADAPTIVE PENALTY GROWTH FACTOR = <real>(1.5)
    [DURING <string list>period_names]
LAGRANGE ADAPTIVE PENALTY REDUCTION FACTOR = <real>(2.0/3.0)
```

```

    [DURING <string list>period_names]
LAGRANGE ADAPTIVE PENALTY THRESHOLD = <real>(0.25)
    [DURING <string list>period_names]
LAGRANGE MAXIMUM PENALTY MULTIPLIER = <real>(100.0)
    [DURING <string list>period_names]
LAGRANGE MULTIPLIER PREDICTION SCALE FACTOR = <real>(0.85)
    [DURING <string list>period_names]
#
# Tolerance Options
LAGRANGE TOLERANCE = <real>(0.0)
    [DURING <string list>period_names]
#
# Miscellaneous Options
LAGRANGE MAXIMUM UPDATES = <integer>(infinity)
    [DURING <string list>period_names]
LAGRANGE NODAL STIFFNESS MULTIPLIER = <real>(0.0)
    [DURING <string list>period_names]
#
# Line search commands
MAXIMUM LINE SEARCH STEP = <real>(2.0)
    [DURING <string list>period_names]
MINIMUM LINE SEARCH STEP = <real>(0.01)
    [DURING <string list>period_names]
#
# Diagnostic output commands, off by default.
ITERATION PLOT = <integer>iter_plot
    [DURING <string list>period_names]
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks
END [CONTROL CONTACT]

```

For a problem containing a contact definition, control contact will be created by default if not defined in the input file. The default control contact will attempt to solve to a relative residual tolerance of $1.0\text{e-}3$ and will exist at level 1 of the solver. If non-default or advanced options are desired the control contact block must be defined directly.

4.5.1 Control Contact Commands

The line commands within the `CONTROL CONTACT` command block are used to control convergence during the contact updates, select the level for the contact control within the multilevel solver, and output diagnostic information. These commands are described in detail in sections [Section 4.5.1.1](#) and [4.1.1](#).

4.5.1.1 Convergence Commands

The command lines listed in this section specify convergence criteria for the contact updates.

```
TARGET RESIDUAL = <real>target_resid
```

```

[DURING <string list>period_names]
TARGET RELATIVE RESIDUAL =
  <real>target_rel_resid(1.0e-3)
[DURING <string list>period_names]
TARGET RELATIVE CONTACT RESIDUAL = <real>target_rel_cont_resid(1.0e-3)
[DURING <string list>period_names]
ACCEPTABLE RESIDUAL = <real>accept_resid
[DURING <string list>period_names]
ACCEPTABLE RELATIVE RESIDUAL = <real>accept_rel_resid(1.0e-2)
[DURING <string list>period_names]
ACCEPTABLE RELATIVE CONTACT RESIDUAL =
  <real>accept_rel_cont_resid(1.0e-1)
[DURING <string list>period_names]
MINIMUM ITERATIONS = <integer>min_iter(0)
[DURING <string list>period_names]
MAXIMUM ITERATIONS = <integer>max_iter(100)
[DURING <string list>period_names]

```

Solver convergence is monitored for control contact in the same way as for the core CG solver. With the exception of two additional command lines used for controlling the residual due to contact, these command lines are the same as those for the core CG solver (described in Section 4.2.1). Here, however, these command lines are applied to the contact control level.

Contact convergence is evaluated by computing the L^2 norm of the residual forces and comparing that with a target convergence criteria specified by the user in the control contact block. The relative residual is computed by dividing the residual by a reference quantity that is indicative of the current loading conditions on the model. Note the residual has units of force (or energy) while the relative residual is a dimensionless quantity where one or larger is an unconverged solution and the value moves towards zero for increasingly exact convergence.

The `TARGET RESIDUAL` command specifies the target convergence residual in terms of the actual residual norm. The `TARGET RELATIVE RESIDUAL` command specifies a convergence criterion in terms of the relative residual. If both command lines are specified, the multilevel solver will accept the contact solution as converged if either the target residual or the target relative residual is below the specified values.

The `TARGET RELATIVE CONTACT RESIDUAL` command line specifies a relative residual tolerance for just the nodes in contact. When this command is used this criterion must be satisfied in addition to the target residual or relative residual for all nodes. The default value for the target relative contact residual is the same value as the target relative residual or 10 times the target residual if the target relative residual has not been specified. As the contact residual is the residual on the subset of nodes of the model currently in contact the contact residual will always be strictly lower than the standard residual. For the `TARGET RELATIVE CONTACT RESIDUAL` to have any effect the value must be set below the `TARGET RELATIVE RESIDUAL`. A small `TARGET RELATIVE CONTACT RESIDUAL` command can be used to give improved confidence in the accuracy of the solution in the contact zone.

Acceptable convergence criteria can also be used for contact convergence. If the solution has not converged to the specified target before the maximum number of iterations is reached, the residual is checked against the acceptable convergence criteria. These criteria are specified via the `ACCEPTABLE RESIDUAL`, `ACCEPTABLE RELATIVE RESIDUAL`, and `ACCEPTABLE RELATIVE CONTACT RESIDUAL` command lines. The concepts of acceptable residual, relative residual, and relative contact residual are the same as those used for the targets. If the solution has not met the target criteria but has met the acceptable criteria, the solution is allowed to proceed. The defaults for each of these acceptable criteria are 10 times the corresponding target criteria defaults.

Control contact will use the same reference type (`EXTERNAL`, `BELYTSCHKO`, `ENERGY`, etc.) as the underlying CG solver block.

The `MAXIMUM ITERATIONS` and `MINIMUM ITERATIONS` command lines specify the maximum and minimum number of contact updates, respectively. The default minimum number of iterations, `min_iter`, is 0. If a number greater than 0 is specified, the multilevel solver will update contact at least that many times, regardless of whether the convergence criteria have been met.

For an explanation of commands common to all controls, see section [4.1.1](#).

4.5.1.2 Tips on Setting Control Contact and CG tolerances

Generally the target residual in the `CG` block should be set to about an order of magnitude below the target residual in the `CONTACT` block, while the acceptable residual in the `CG` block should be set to a very large number. The acceptable should be large enough to assure solution acceptance for any remotely reasonable result.

For each control contact step, a set of active contact constraints is held fixed. The core `CG` solver subsequently solves the analysis problem subject to a linearized version of those active contact constraints. Occasionally control contact will pick a set of constraints that cannot be solved; for example, creating two constraints that conflict such that the solution is over determined. When this happens, the `CG` solver will never be able to drop the residual of the over-constrained system. In this case, contact needs to try again to pick a better constraint set. When `CG` hits the maximum number of iterations and meets the high acceptable tolerance, it will return to the contact solver. Though the residual is still high, it usually has enough information to at least help contact select an improved constraint set. Note, when using control contact the actual true tolerance of the solution is the control contact residual. The `CG` residual is used for the intermediate model problems. Thus, if control contact eventually converges to a reasonably small residual, the solution itself is correct even if some of the intermediate `CG` steps only converged on a high acceptable residual.

4.5.1.3 Augmented Lagrange Initialization Commands

```
LAGRANGE INITIALIZE = NONE|MULTIPLIER|PENALTY|BOTH (MULTIPLIER)
[DURING <string list>period_names]
```

The `LAGRANGE INITIALIZE` command is used to control how Lagrange multipliers are set at the beginning of a load step. The default value of `MULTIPLIER` specifies that the Lagrange multipliers will be initialized to some factor of the previous step Lagrange multipliers. By default this multiplication factor is 0.85 and can be changed with the `LAGRANGE MULTIPLIER PREDICTION`

SCALE FACTOR command.

The `NONE` option turns off initialization of the Lagrange multipliers, setting them to zero. This option may aid convergence in some very nonlinear analyses where the next load step is in a completely different contact state than the previous load step. The `PENALTY` option initializes Lagrange multipliers based off of material stiffness and gap. The `BOTH` initializes Lagrange multipliers based off of both the old multiplier and the the material stiffness and gap in an attempt to achieve a more efficient prediction. It is generally recommended that only the `MULTIPLIER` or `NONE` options be used as the penalty based initialization is somewhat experimental at this time.

4.5.1.4 Augmented Lagrange Adaptive Penalty Commands

```
# Adaptive Penalty Options
LAGRANGE ADAPTIVE PENALTY = OFF|SEPARATE|UNIFORM (UNIFORM)
[DURING <string list>period_names]
LAGRANGE ADAPTIVE PENALTY GROWTH FACTOR = <real>(1.5)
[DURING <string list>period_names]
LAGRANGE ADAPTIVE PENALTY REDUCTION FACTOR = <real>(2.0/3.0)
[DURING <string list>period_names]
LAGRANGE ADAPTIVE PENALTY THRESHOLD = <real>(0.25)
[DURING <string list>period_names]
LAGRANGE MAXIMUM PENALTY MULTIPLIER = <real>(100.0)
[DURING <string list>period_names]
LAGRANGE MULTIPLIER PREDICTION SCALE FACTOR = <real>(0.85)
[DURING <string list>period_names]
```

The command lines listed above can be used to control the augmented Lagrange contact adaptive penalty algorithm. Augmented Lagrange contact enforcement is activated when `enforcement = al` is set in the contact definition block.

The `LAGRANGE ADAPTIVE PENALTY` command indicates how the penalty multiplier will be adjusted as the problem progresses. The option `SEPARATE` scales the penalty differently for each contact interaction. The option `UNIFORM` scales all interactions equally. The default option is `UNIFORM`.

The `LAGRANGE ADAPTIVE PENALTY GROWTH FACTOR` command sets the adaptive penalty growth factor, while the `LAGRANGE ADAPTIVE PENALTY REDUCTION FACTOR` command sets the adaptive penalty reduction factor. The `LAGRANGE ADAPTIVE PENALTY THRESHOLD` sets the adaptive penalty threshold. The `LAGRANGE MAXIMUM PENALTY MULTIPLIER` sets the maximum penalty multiplier, which is the upper bound on the penalty multiplier when running augmented Lagrange contact.

In augmented Lagrange contact, if the current gap magnitude is greater than the previous gap magnitude, the penalty multiplier is reduced by multiplying it by the penalty reduction factor. If the current gap magnitude is between the previous gap magnitude and the previous gap magnitude times the penalty threshold, the penalty multiplier is increased by multiplying it by the penalty growth factor. The lower bound for the penalty multiplier is 1.0 and the upper bound is the maximum penalty multiplier.

The `LAGRANGE MULTIPLIER PREDICTION SCALE FACTOR` sets a scaling factor on the Lagrange multipliers when moving from one load step to the next. Generally the converged Lagrange multipliers at a new load step will be close to the multipliers obtained in the previous load step. When predicting the next configuration, a default scaling factor of 0.85 is applied to the previous Lagrange multipliers to reduce the danger of overshooting the contact solution. A value closer to 1.0 may accelerate convergence but may also introduce issues by overshooting the true contact solution.

4.5.1.5 Augmented Lagrange Tolerance

```
# Tolerance Options
LAGRANGE TOLERANCE = <real>(0.0)
[DURING <string list>period_names]
```

The `LAGRANGE TOLERANCE` can be used to set the Lagrange multiplier tolerance for updating. Once the change to the Lagrange multiplier is less than the tolerance, the Lagrange multiplier will no longer be updated. The default is 0.0, which always updates the Lagrange multiplier.

4.5.1.6 Augmented Lagrange Miscellaneous Options

```
# Miscellaneous Options
LAGRANGE MAXIMUM UPDATES = <integer>(infinity)
[DURING <string list>period_names]
[DURING <string list>period_names]
LAGRANGE NODAL STIFFNESS MULTIPLIER = <real>(0.0)
[DURING <string list>period_names]
```

The `LAGRANGE MAXIMUM UPDATES` command sets the maximum number of augmented Lagrange updates performed during a load step. By default, the Lagrange multiplier is always updated. Setting this to zero makes contact a pure penalty formulation.

The `LAGRANGE NODAL STIFFNESS MULTIPLIER` sets a scale factor on the amount of contact stiffness included in the nodal preconditioner. The default value is 0.0, which means no contact stiffness is present in the nodal preconditioner. A value of 1.0 distributes the full contact stiffness to the contact nodes and a value much greater than 1.0 effectively pins (or fixes) the contact nodes in the nodal preconditioner.

4.5.2 Augmented Lagrange Control Contact

For augmented Lagrange enforcement, a search is only performed once in the predicted configuration of a new load step. It is crucial that the predicted configuration be reasonable. Experience has shown that no prediction is often best, so it is recommended that the load step predictor be turned off when using augmented Lagrange enforcement for contact. This can be accomplished by setting the scale factor predictor to zero (See Section [4.9.1](#)).

The contact search provides a set of all possible interactions. In the log file example in Section [4.5.2.1](#), the listed `NUM INTERACTIONS` is the number of possible interactions that were found

in the contact search. Each interaction is a node-face or face-face pair, depending on the constraint formulations used in the contact definition.

Interactions can exist in one of three states: RELEASED, CAPTURED, or DUBIOUS. Tied interactions are determined once during the search and can only be CAPTURED or RELEASED. Other interactions start in a DUBIOUS or RELEASED state after the search. To reduce chatter, these interactions can only move to CAPTURED or RELEASED by moving through the DUBIOUS state.

The DUBIOUS state serves as an intermediate state. During a model problem, interactions in the DUBIOUS state are enforced with a nonlinear penalty kernel during the model problem, while interactions in the CAPTURED state use a linear penalty kernel. This is why the first model problem is often the most difficult. The logic to move interactions between states is based on the gapping history combined with the value of the accumulated Lagrange multiplier.

During a contact update, the Lagrange multiplier is augmented before the state determination by adding the penalty kernel to the Lagrange multiplier. The maximum relative change in the value of the Lagrange multiplier is listed in the log file as RELATIVE LMULT CHANGE. Because the convergence is determined exclusively on the residual balance, it is sometimes desirable to set a LAGRANGE TOLERANCE so that if the change in the Lagrange multiplier is small enough, the update is not done. This preserves the residual balance determined in the model problem, assuming there is no change in the interaction states.

As contact iterations progress, the values of MAX GAP, MAX RELATIVE GAP and RELATIVE LMULT CHANGE should decrease and the set of interactions should stop changing (signified by the output line NO ACTIVE SET CHANGE). Successive iterations should report a RELATIVE RESIDUAL for control contact closer to the RELATIVE RESIDUAL reported in the model problem until convergence is reached.

4.5.2.1 Augmented Lagrange Control Contact Log File Example

```
=====
Begin load step =      37 Solution period Adagio_Procedure_time_control_1 is 14.8% complete
  Old Time          Time Step          New Time          Stop Time          CPU Time(s)          Wall Time(s)
  7.4000e-03        2.0000e-04        7.6000e-03        5.0000e-02        3.8784e+00        4.0469e+00
=====
                                MP          RELATIVE          EXTERNAL
                                ITER          RESIDUAL          RESIDUAL          REFERENCE          ENERGY DISPLACEMENT
=====
                                0          9.358e-03        6.885e-02        1.359e-01          -          -
                                16          1.814e-06        1.595e-05<T        1.137e-01        4.634e-17        8.057e-08
=====
MAX GAP              = 2.779e-05 PREVIOUS = 0.000e+00
MAX RELATIVE GAP     = 6.175e-03 PREVIOUS = 0.000e+00
NUM INTERACTIONS     = 60
RELEASED INTERACTIONS = 0
CAPTURED INTERACTIONS = 60
DUBIOUS INTERACTIONS = 0
RELATIVE LMULT CHANGE = 1.465e-01
ACTIVE SET CHANGE
=====
CONTACT ITERATION =      0
ABSOLUTE RESIDUAL =    3.444e-03
RELATIVE RESIDUAL =    2.914e-02
=====
... ( Skipped Contact Iteration 1 - 9 for readability ) ...
=====
                                MP          RELATIVE          EXTERNAL
                                ITER          RESIDUAL          RESIDUAL          REFERENCE          ENERGY DISPLACEMENT
=====
```

	0	8.104e-05	6.810e-04	1.190e-01	-	-
	5	1.295e-06	1.088e-05<T	1.190e-01	9.540e-19	2.180e-08

```

MAX GAP           = 1.136e-06 PREVIOUS = 1.129e-06
MAX RELATIVE GAP  = 1.514e-04 PREVIOUS = 1.506e-04
NUM INTERACTIONS  = 60
RELEASED INTERACTIONS = 4
CAPTURED INTERACTIONS = 56
DUBIOUS INTERACTIONS = 0
RELATIVE LMULT CHANGE = 7.380e-03
NO ACTIVE SET CHANGE

```

```

CONTACT ITERATION = 10
ABSOLUTE RESIDUAL = 5.179e-05
RELATIVE RESIDUAL = 4.352e-04<T

```

LEVEL 1				ITERATIONS	TOTAL
CONTACT	MODEL PROB	LOAD STEP	AVERAGE	TOTAL	CPU (s)
CONVERGED	11	114	10	4069	4.825e+00

4.5.3 Kinematic Control Contact

For kinematic enforcement, a search is performed each contact iteration to provide a set of nodes in contact. This constraint set is held constant during the subsequent model problem. For frictional contact, slave nodes are fixed to master faces during the CG iterations. For frictionless contact, the slave nodes are fixed in the normal direction but are allowed to slide during the CG iterations. After model problem convergence, the constraint set is updated to reflect the changing contact conditions. This update gives rise to a force imbalance to be solved in another model problem.

Changing or updating the constraint set is called a contact update. Multiple contact updates are typically required before equilibrium is achieved. The contact update includes a search, a gap removal, an equilibrium query, and a slip calculation if equilibrium is not satisfied.

New constraints are detected in the search phase based on the current deformed configuration of the model. If a node is found to penetrate a master surface, the node is added to the set of constraints.

The gap removal phase involves creating or destroying constraints and calculating push-back vectors for slave nodes. A penetration may be removed at once (the default behavior), or it may be incrementally removed over a number of model problems. For interactions where contacting surfaces are free to separate, a slave node constraint continues to exist as long as there is a compressive force between the slave node and the master surface. The constraint changes during the gap removal phase to reflect changes in the shape and orientation of the master surface. Constraints are destroyed when a tensile force exceeding a specified tolerance (see Section 8.6.2.6) exists at the master/slave interface.

During the slip portion of the contact update, a residual force is calculated at each node and resolved into normal and tangential components. This force reflects changes in residual due to gap removal that occurred earlier in the update. For frictional contact, the friction coefficient is used to determine the tangential load capacity. If the tangential loads exceed this capacity, nodes will slip.

The procedure for kinematic control contact is illustrated in the following sequence of figures. It is assumed in these figures that control contact is the level 1 solver, so the model problems are directly solved by the core solver. In Figure 4.2, the model problem has three constraints that are enforced during iterations of the core solver. However, during solution of the model problem an additional node penetrates the master surface.

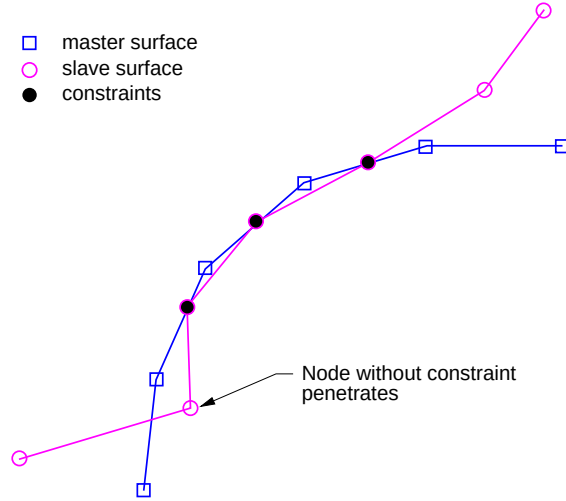


Figure 4.2: Contact configuration at the beginning of the contact update.

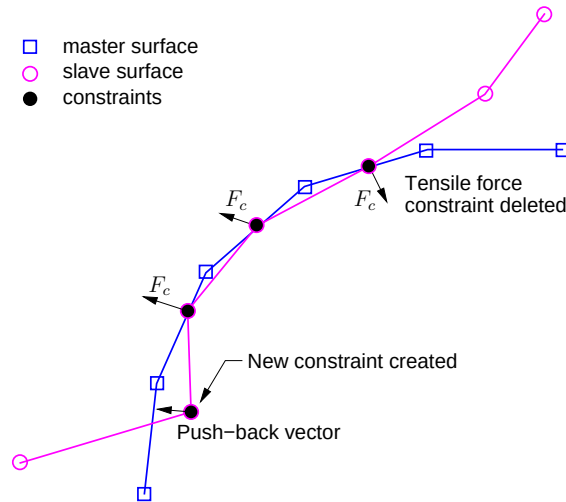


Figure 4.3: Contact gap removal (after contact search).

Once the model problem has converged, a contact update is performed. During the search, one node is found to have penetrated the surface, resulting in a new constraint for this node. A push-back vector is calculated for this node to remove the penetration. In the gap removal portion of the update (Figure 4.3), one of the constraints is found to have accumulated a tensile force large enough that surfaces should separate, indicating that this constraint should be eliminated.

In the second phase of the contact update, slip is allowed to occur along the master/slave interface. After the gaps are removed, the external and internal force vectors are recomputed, and forces are partitioned along normal and tangential directions of the master surface. This process is illustrated in Figure 4.4.

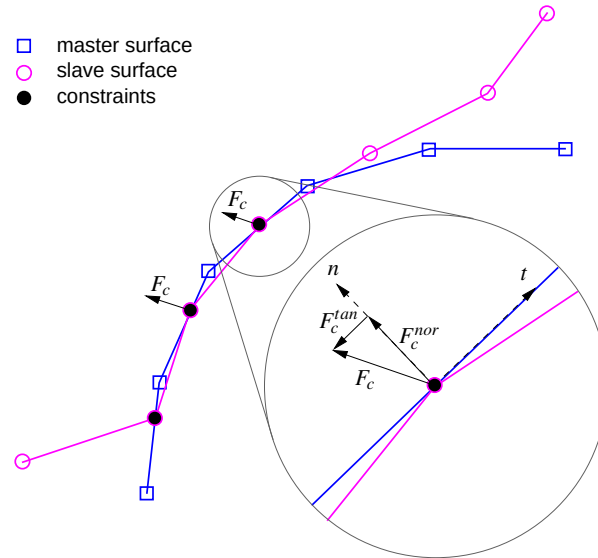


Figure 4.4: Contact slip calculations.

4.5.4 CG with Control Contact Input Example

```

begin solver
  begin control contact
    target relative residual = 1.0e-4
    maximum iterations      = 100
  end
  begin cg
    acceptable relative residual = 1.0e10
    target relative residual = 1.0e-6
    maximum iterations      = 100
    iteration print         = 25
    begin full tangent preconditioner
      minimum smoothing iterations = 25
    end
  end
end
end

```

4.6 Control Stiffness

Control stiffness is an augmented Lagrange iterative solution strategy for solving mechanics models that have *widely* varying stiffness in various modes of material response. For instance, nearly incompressible materials have a bulk/volumetric response that is much stiffer than the corresponding shear/deviatoric response. A similar case arises for materials that are orthotropic or anisotropic in nature and exhibit widely varying stiffness in the principal material directions.

Using control stiffness allows part of the constitutive response of a material to be softened or stiffened to give a tangent response that results in model problems that can be solved more easily by the core solver. This is accomplished by scaling a chosen stress increment up or down in a given model problem and adding it to a saved reference stress to yield a scaled stress that is used for equilibrium evaluations. The reference stress accumulates the model problem stress increments so that the true material response for a time step is obtained after solving a sequence of model problems.

This capability is highly specialized to certain use cases and is not recommended for most problems. It is also only available for use with certain specialized material models. Use of this capability should be considered only as a last resort when convergence cannot be achieved in any other way. Potential users of this feature are advised to first contact the Sierra/SM development team for advice on how to effectively use this capability.

The command block for control stiffness is as follows:

```
BEGIN CONTROL STIFFNESS [<string>stiffness_name]
#
# convergence commands
TARGET <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT
    = <real>target [DURING <string list>period_names]
TARGET RELATIVE <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT|
    STRAIN INCREMENT
    = <real>target_rel [DURING <string list>period_names]
ACCEPTABLE <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT
    = <real>accept [DURING <string list>period_names]
ACCEPTABLE RELATIVE <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT|
    STRAIN INCREMENT
    = <real>accept_rel [DURING <string list>period_names]
REFERENCE = EXTERNAL|INTERNAL|BELYTSCHKO|RESIDUAL|ENERGY (EXTERNAL)
    [DURING <string list>period_names]
MINIMUM ITERATIONS = <integer>min_iter(0)
    [DURING <string list>period_names]
MAXIMUM ITERATIONS = <integer>max_iter
    [DURING <string list>period_names]
#
```

```

# level selection command
LEVEL = <integer>stiffness_level
#
# diagnostic output commands, off by default.
ITERATION PLOT = <integer>iter_plot
    [DURING <string list>period_names]
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks
END [CONTROL STIFFNESS <string>stiffness_name]

```

To enable control stiffness, a `CONTROL STIFFNESS` command block must exist in the `SOLVER` command block. The command lines within the `CONTROL STIFFNESS` command block are used to control convergence during the control stiffness updates, select the level for control stiffness within the multilevel solver, and output diagnostic information. These commands are described in detail in sections Section [4.6.1](#) and [4.1.1](#).

4.6.1 Convergence Commands

The command lines listed in this section are placed in the `CONTROL STIFFNESS` command block to control convergence criteria for stiffness control within the multilevel solver.

```

TARGET <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT
    = <real>target [DURING <string list>period_names]
TARGET RELATIVE <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT|STRAIN INCREMENT
    = <real>target_rel [DURING <string list>period_names]
ACCEPTABLE <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT
    = <real>accept [DURING <string list>period_names]
ACCEPTABLE RELATIVE <string>RESIDUAL|PRESSURE INCREMENT|
    SDEV INCREMENT|STRESS INCREMENT|STRAIN INCREMENT
    = <real>accept_rel [DURING <string list>period_names]
REFERENCE = EXTERNAL|INTERNAL|BELYTSCHKO|RESIDUAL|ENERGY (EXTERNAL)
    [DURING <string list>period_names]
MINIMUM ITERATIONS = <integer>min_iter(0)
    [DURING <string list>period_names]
MAXIMUM ITERATIONS = <integer>max_iter
    [DURING <string list>period_names]

```

Convergence for problems using control stiffness requires that two criteria be met. The first criterion is that force equilibrium must be achieved. The second criterion is that the scaled stress used for equilibrium must be close enough to the unscaled stress that is determined from the unmodified material model.

The command lines listed above for the equilibrium evaluation are similar to those used for the core CG solver and have similar meaning. Here, the command lines are applied as part of determining convergence for the control stiffness series of model problems. Some of the command lines have

multiple options, e.g., the command line beginning with `TARGET` has five options. This practice has been used for the command lines beginning with `TARGET`, `TARGET RELATIVE`, `ACCEPTABLE`, and `ACCEPTABLE RELATIVE`. That all of these command lines can be appended with the `DURING` specification, as discussed in Section 4.2.1.

Convergence of equilibrium using scaled stresses is measured by computing the L^2 norm of the residual and comparing that residual norm with target convergence criteria specified by the user. Convergence can be monitored either directly in terms of the residual norm, or in terms of a relative residual, which is the residual norm divided by a reference quantity that is indicative of the current loading conditions on a model. Either the residual, the relative residual, or both of these can be used for checking convergence at the same time. The `TARGET RESIDUAL` command line specifies the target convergence criterion in terms of the actual residual norm. The `TARGET RELATIVE RESIDUAL` command line specifies a convergence criterion in terms of the relative residual. If both absolute and relative criteria are specified for the residual calculation, equilibrium is accepted if either criterion is met.

The multilevel solver also allows acceptable convergence criteria to be input for residual convergence. If the solution has not converged to the specified targets before the maximum number of iterations is reached, the residual is checked against the acceptable convergence criteria. These criteria are specified via the `ACCEPTABLE RESIDUAL` and `ACCEPTABLE RELATIVE RESIDUAL` command lines. The concepts of absolute and relative values are the same here as discussed for the target limits. When a multilevel solver is being used, the level 0 (cg) acceptable residual and relative residual defaults are overridden and set to $1e20$. This choice was made to improve the overall robustness of the multilevel solver when faced with difficult contact problems. As long as the targets in the controls (contact, stiffness, failure, or damped solve) are met, then the solution is converged and allowed to continue time stepping.

If relative residuals are given in the convergence criteria, the `REFERENCE` command line can be used to select the method for computing the reference load. This command line has several options: `EXTERNAL`, `INTERNAL`, `BELYTSCHKO`, `RESIDUAL` and `ENERGY`. When the `EXTERNAL` option, which is the default, is selected, the reference load is computed by taking the L^2 norm of the current external load. The `INTERNAL` option uses the norm of the internal forces as the reference load. The `BELYTSCHKO` option uses the maximum of the norm of the external load, the norm of the reactions, and the norm of the internal forces. The `RESIDUAL` option denotes that the residual from the initial residual should be used as the reference quantity. The `ENERGY` residual is the energy conjugate norm of the residual and reference vectors. These criteria are further explained in Section 4.2.1.

For control stiffness, unlike other controls, the convergence of the series of model problems to obtain the final solution for a time step is also based on other criteria that measure how far apart the scaled and unscaled stress responses are from each other. Either a direct error measure of the difference of the scaled and unscaled stress is used or a relative strain error computed using the difference between the scaled and unscaled stress and a representative modulus is employed.

There are many variants of the `TARGET` command that can be used for the direct stress differences. The `TARGET PRESSURE INCREMENT` and `TARGET SDEV INCREMENT` commands are used to specify convergence for nearly incompressible materials based on the pressure and deviatoric stress increments, respectively. Also, `TARGET STRESS INCREMENT` is used to base convergence on

the stress increment for materials which have their entire behavior softened as part of a control stiffness approach. If desired, the `RELATIVE` form of these convergence criteria can be used. If the `TARGET RELATIVE STRAIN INCREMENT` command is specified, the strain error measure between the scaled and unscaled stress will be computed and used.

Any combination of these criteria can be specified. If more than one is specified, all the criteria must be satisfied. However, each material block contributes either to the direct stress error measures or to the relative strain error measure. If a material block specifies a positive non-zero `REFERENCE STRAIN` in its definition, it will contribute to the relative strain error measure. Otherwise, it will contribute to the direct stress error measures. Finally, acceptable convergence criteria can be input for converging the difference between the scaled and unscaled stress. If the solution has not met the target criteria for equilibrium and the difference between the scaled and unscaled stresses but meets the acceptable criteria, it is allowed to proceed to the next time step.

The `MAXIMUM ITERATIONS` and `MINIMUM ITERATIONS` command lines specify the maximum and minimum number of control stiffness updates, respectively. The default minimum number of iterations, `min_iter`, is 0. If a number greater than 0 is specified for `min_iter`, the multilevel solver will update stiffness at least that many times, regardless of whether the convergence criteria have been met.

For an explanation of commands common to all controls, see section [4.1.1](#).

Separate from the iteration plot commands, control stiffness provides several other element variables for diagnostic output.

- `bulk_strain_error`
- `shear_strain_error`
- `axial_strain_error`
- `anisotropic_strain_error`

4.7 Control Failure

Control failure is used to improve the conditioning and accuracy of models that involve element death. To use control failure, the an element death block (as defined in Section 6.5) and a multilevel solver control failure must both be defined.

When element death is used in the absence of control failure at the end of a load step all elements that meet the element death criteria are removed and the analysis moves on to the next step. There are two issues with this approach control failure will remedy. First death of one element may increase the load on remaining elements which should cause additional element failures. Second death of one element may relieve the load on another element, possibly putting that element below the failure criteria. Thus element death without control failure could either under predict or over predict the actual number of elements that should be removed in a given time step.

Control failure with material criterion limits the failure only to the single most critical element during each control failure iteration. With control failure when an element dies the model problem will be repeated with the element in a failed state in the presence of control failure. Iterations of the control failure continue until no additional elements are marked for failure.

The command block for control failure is as follows:

```
BEGIN CONTROL FAILURE [<string>failure_name]
#
#  convergence control command
MAXIMUM ITERATIONS = <integer>max_iter
    [DURING <string list>period_names]
#
#  level selection command
LEVEL = <integer>failure_level
#
#  diagnostic output commands, off by default.
ITERATION PLOT = <integer>iter_plot
    [DURING <string list>period_names]
ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks
END [CONTROL FAILURE <string>failure_name]
```

To enable control failure, a `CONTROL FAILURE` command block must exist in the `SOLVER` command block. The command lines within the `CONTROL FAILURE` command block are used to establish convergence criteria, select the level for control failure within the multilevel solver, and output diagnostic information. These commands are described in detail in sections Section 4.7.1 and 4.1.1.

Convergence of control failure is defined as having no new elements have been marked as failed in the previous model problem. Control failure will also complete if it reaches the `MAXIMUM ITERATIONS` number of model problem evaluations in a single load step.

When element death is not controlled by a material criterion, all elements that satisfy the specified criterion are marked at the end of the model problem.

When a material criterion is used, the material will set the variable “failure_flag” to 1.0 when the failure criterion is met. At the end of the model problem, control failure will set the “failure_flag” of the failed element with the largest “failure_measure” to 2.0, thereby marking it. For the marked element, at the beginning of the following model problem the material model will set the “failure_flag” to 3.0 as it begins decaying the material, while all other instances of “failure_flag” with a value of 1.0 are set back to 0.0. Control failure iterations continue until no elements are marked for failure (i.e. no elements have a “failure_flag” of 2.0) or the `MAXIMUM ITERATIONS` is reached. What this means is that only the most critical element is removed each model problem. This one by one element removal prevents potential over prediction of failure if several elements exceed the a failure criterion on the same model problem.

4.7.1 Convergence Command

The only user input convergence criteria that is allowed for control failure is specification of maximum iterations.

```
MAXIMUM ITERATIONS = <integer>max_iter  
[DURING <string list>period_names]
```

Iterations for control failure will continue until no new elements are failing unless the user specified number of maximum iterations is reached. If a maximum number of iterations is specified in the input, control failure iterations will only continue until this number is reached. If the maximum number of iterations is reached, the analysis will terminate in an unconverged state and with an explanatory message in the log file. This command can be appended with the `DURING` specification, as discussed in Section 4.2.1. The unconverged state can also trigger adaptive time stepping if it is active.

4.7.2 Level Command

When used in a multilevel solver the `CONTROL FAILURE` command block should be processed at the highest solver level and should be the only control acting at that level. For more details on the `LEVEL` command see Section 4.1.1.

4.7.3 Other Commands

For an explanation of commands related to iteration plot, see section 4.1.1.

4.7.4 Control Failure Input Example

The following example shows the setup for a general multilevel solver involving control contact and control failure. Each load step first a multilevel control contact model problem is solved. Control failure then removes the most failed element based off of the material failure criterion. Control contact again solves the model problem with that element removed. This process continues until no new elements fail. Due to the large number of iterative resolves required the accuracy provided by control failure for such a situation may come at a substantial run time cost.

```
begin element death
  include all blocks
  material criterion = on
end

Begin solver
  Begin control failure
    level = 2
  end control failure

  Begin control contact
    level = 1
    target relative residual = 1.0e-6
  end control contact

  Begin cg
    target relative residual = 1.0e-8
  end cg
End
```

4.8 Control Damped Solve

Control damped solve is used to aid implicit quasistatic runs in convergence. With implicit quasistatic problems, it is sometimes very difficult to achieve convergence with the cg solver alone (even when used in combination with the FETI equation solver). Some problems where convergence is difficult to achieve includes: problems with unconstrained rigid body motion, problems that contain very nonlinear physics, implicit contact problems that involve rebound or bounce, implicit contact problems where sliding occurs, and implicit fracture problems where failure introduces rigid body motion or softening materials.

At a high level, `CONTROL DAMPED SOLVE` places artificial springs to ground on nodes to allow the problem to converge. Once the springs are placed, `CONTROL DAMPED SOLVE` will iterate until the artificial spring force is below the user defined tolerance. To achieve convergence, the ground position of the springs is moved to the last predicted position at the control level. If the next predicted position is the same, the spring applies no force.

The command block for control damped solve is as follows:

```
BEGIN CONTROL DAMPED SOLVE [<string>damped_solve_name]
  # Convergence commands
  TARGET SPRING FORCE = <real>target_spring_force
  TARGET RELATIVE SPRING FORCE = <real>target_rel_spring_force
  ACCEPTABLE SPRING FORCE = <real>accept_spring_force
  ACCEPTABLE RELATIVE SPRING FORCE = <real>accept_rel_spring_force
  REFERENCE = EXTERNAL|INTERNAL(EXTERNAL)
  MAXIMUM ITERATIONS = <integer>max_iter
  MINIMUM ITERATIONS = <integer>min_iter
  #
  # level selection command
  LEVEL = <integer>damped_solve_level
  #
  # artificial spring commands
  STIFFNESS FORM = SHIFTED|SCALED(SCALED)
  STIFFNESS FACTOR = <REAL>stiffness_factor
  FORCE DECAY = NONE|ITERFRAC|HALVED|TENTHED(HALVED)
  EVERY NTH NODE = <integer>nth_node(1)
  #
  # diagnostic output commands, off by default.
  ITERATION PLOT = <integer>iter_plot
    [DURING <string list>period_names]
  ITERATION PLOT OUTPUT BLOCKS = <string list>plot_blocks
END [CONTROL DAMPED SOLVE <string>damped_solve_name]
```

To enable control damped solve, a `CONTROL DAMPED SOLVE` command block must exist in the `SOLVER` command block. The command lines within the `CONTROL DAMPED SOLVE` command block are used to establish convergence criteria, select the level for control damped solve within the multilevel solver, specify characteristics of the artificial springs, and output diagnostic information. These commands are described in detail in sections [Section 4.8.1](#) and [4.1.1](#).

4.8.1 Convergence Command

The command lines listed in this section specify convergence criteria for the control damped solve level.

```
TARGET SPRING FORCE = <real>target_spring_force
TARGET RELATIVE SPRING FORCE = <real>target_rel_spring_force
ACCEPTABLE SPRING FORCE = <real>accept_spring_force
ACCEPTABLE RELATIVE SPRING FORCE = <real>accept_rel_spring_force
REFERENCE = EXTERNAL|INTERNAL(EXTERNAL)
MAXIMUM ITERATIONS = <integer>max_iter
MINIMUM ITERATIONS = <integer>min_iter
```

Solver convergence is monitored for control damped solve by examining the force in the artificial springs. Convergence is evaluated by computing the L^2 norm of the spring forces and comparing that with a target convergence criteria specified by the user in the control damped solve block. The relative spring force is computed by dividing the spring force by a reference quantity that is indicative of the current loading conditions on the model. Note the spring force has units of force while the relative spring force is a dimensionless quantity where one or larger is an unconverged solution and the value moves towards zero for increasingly exact convergence. When using CONTROL DAMPED SOLVE, at least one of the following must be specified: TARGET SPRING FORCE, TARGET RELATIVE SPRING FORCE, ACCEPTABLE SPRING FORCE, or ACCEPTABLE RELATIVE SPRING FORCE.

The TARGET SPRING FORCE command specifies the target convergence force in terms of the actual spring force norm. The TARGET RELATIVE SPRING FORCE command specifies a convergence criterion in terms of the relative spring force norm. If both command lines are specified, the multilevel solver will accept the damped solve solution as converged if either the target spring force norm or the target relative spring force norm is below the specified values.

Acceptable convergence criteria can also be used for damped solve convergence. If the solution has not converged to the specified target before the maximum number of iterations is reached, the spring force is checked against the acceptable convergence criteria. These criteria are specified via the ACCEPTABLE SPRING FORCE and ACCEPTABLE RELATIVE SPRING FORCE command lines. The concepts of acceptable spring force and acceptable relative spring force are the same as those used for the targets. If the solution has not met the target criteria but has met the acceptable criteria, the solution is allowed to proceed. The defaults for each of these acceptable criteria are 10 times the corresponding target criteria defaults.

If relative spring forces are given in the convergence criteria, the REFERENCE command line can be used to select the method for computing the reference load. This command line currently has two options: EXTERNAL and INTERNAL. When the EXTERNAL option, which is the default, is selected, the reference load is computed by taking the L^2 norm of the current external load. The INTERNAL option uses the norm of the internal forces as the reference load. These criteria are further explained in Section 4.2.1.

The MAXIMUM ITERATIONS and MINIMUM ITERATIONS command lines specify the maximum and minimum number of damped solve updates, respectively. The default minimum number of

iterations, `min_iter`, is 0. If a number greater than 0 is specified, the multilevel solver will update the damped solve at least that many times, regardless of whether the convergence criteria have been met.

For an explanation of commands common to all controls, see section [4.1.1](#).

4.8.2 Level Command

When used in a multilevel solver the `CONTROL DAMPED SOLVE` command block is recommended to be at level 1 and be the only control acting at that level. This allows the artificial spring force to be minimized before considering other controls. For more details on the `LEVEL` command see Section [4.1.1](#).

4.8.3 Control Damped Solve Artificial Spring Commands

```
STIFFNESS FORM = SHIFTED|SCALED(SCALED)
STIFFNESS FACTOR = <REAL>stiffness_factor
FORCE DECAY = NONE|ITERFRAC|HALVED|TENTHED(HALVED)
EVERY NTH NODE = <integer>nth_node(1)
```

Commands directly relating to the artificial springs created by `CONTROL DAMPED SOLVE` are described in this section. `STIFFNESS FORM` specifies how the stiffness of the artificial springs is determined. If option `SHIFTED` is used all artificial springs added have the same stiffness. That spring's stiffness is the maximum value of the nodal stiffness from the element stiffness contributions then scaled by the `STIFFNESS FACTOR`. If the option `SCALED` is used the artificial springs stiffness is the node's nodal stiffness scaled by the `STIFFNESS FACTOR`.

The `STIFFNESS FACTOR` command is used to specify the scale factor for which to scale the spring stiffness. This scale factor can be between 0.0 and 1.0.

The `FORCE DECAY` command is used to decrease the stiffness used in the internal force routine (but not the preconditioner) with each control level iteration. The option `NONE` does nothing to the stiffness. The option `ITERFRAC` decreases the stiffness by 1 over the current iteration number. The option `HALVED` decreases the stiffness by half with each iteration. The option `TENTHED` decreases the stiffness by a factor of 10 with each iteration. The current `FORCE DECAY FACTOR` is echoed in the log file.

The `EVERY NTH NODE` command is used to specify the nodes for which the artificial springs are applied to. If `EVERY NTH NODE = 1`, springs will be applied to every node in the model. If `EVERY NTH NODE = 5`, springs will be applied to every 5th node in the model and then perturbed by 1 every `CONTROL DAMPED SOLVE` iteration.

Currently, there is not an automated method of deciding which nodes need springs. More advanced options for spring placement are currently in-development.

When a `CONTROL DAMPED SOLVE` block is included in the input deck, an integer field named `Control_NodeMask` is created. A node with an integer value of `Control_NodeMask` greater than 0 will have a spring added to ground. This field is what Sierra/SMuses to determine which nodes should have artificial springs. If desired, a user can set this field manually either by using a

script like `exodus.py` or by setting the field with an initial condition block. To keep the field at the user defined value set `EVERY NTH NODE = 0` to avoid the internals from resetting the field.

4.8.4 Other Commands

For an explanation of commands related to iteration plot, see section [4.1.1](#).

4.8.5 Control Damped Solve Input Example

The following example shows the setup for a general multilevel solver involving control contact and control damped solve. Each load step first a multilevel control damped solve model problem is solved. Once the control damped solve force converges to the specified value, control contact is then solved at the next control level. Control damped solve again solves the model problem with the contact forces included, and the process continues until both levels converge to their specified criteria.

```
Begin solver

  Begin cg
    target relative residual = 1.0
  end cg

  Begin control damped solve
    level = 1
    target relative spring force = 1.0e-3
    stiffness shift factor = 1.0e-2
    maximum iterations = 100
  end control damped solve

  Begin control contact
    level = 2
    target relative residual = 1.0e-6
  end control contact

End
```

4.9 Predictors

Predictors are an important component of the nonlinear implicit solution strategy in Sierra/SM. A predictor is used to generate an initial trial solution for a load step or for a multilevel solver model problem. The goal of the predictor is to generate a trial solution that is as close as possible to the actual converged solution. A good prediction that gives a trial solution close to the actual solution can dramatically reduce the number of iterations required for convergence.

There are two types of predictors used in Adagio: the load step predictor and the level 1 predictor. The load step predictor, documented in Section 4.9.1, estimates a solution at the beginning of a new load step, and is applicable to all types of models. The level 1 predictor estimates the solution at the beginning of a new level 1 model problem in the multilevel solver. This predictor type, documented in Section 4.9.2, is only applicable for the multilevel solver.

4.9.1 Loadstep Predictor

```
BEGIN LOADSTEP PREDICTOR
  TYPE = <string>SCALE_FACTOR|SECANT|
        EXTERNAL|EXTERNAL_FIRST (SECANT)
  SCALE FACTOR = <real>scale_factor(1.0)
                [<real>first_scale_factor]
                [DURING <string list>period_names]
  SLIP SCALE FACTOR = <real>slip_factor(1.0)
                    [DURING <string list>period_names]
END [LOADSTEP PREDICTOR]
```

The `LOADSTEP PREDICTOR` command block controls the behavior of the predictor that is used to predict the solution at the beginning of a new load step. This command block is placed in the `SOLVER` scope.

4.9.1.1 Predictor Type

```
TYPE = <string>SCALE_FACTOR|SECANT|
      EXTERNAL|EXTERNAL_FIRST (SECANT)
```

There are multiple types of load step predictors available in Adagio that are suitable for different types of analysis. The predictor type is selected using the `TYPE` command line in the `LOADSTEP PREDICTOR` command block.

- The scale factor predictor is selected with the `SCALE_FACTOR` option. This type of predictor extrapolates from the solution of the previous time step using the velocity field from that step, multiplied by a scale factor, to obtain a trial solution. The actual scale factor that is applied can be controlled with the `SCALE FACTOR` command line, which also appears in the `LOADSTEP PREDICTOR` command block. This type of predictor works well with models that have smoothly varying loads.
- The secant predictor is selected with the `SECANT` option, which is useful for scenarios in which the loading may not be monotonic. This type of predictor uses a line search to compute an optimal scaling factor for use in extrapolating from the previous solution. This type of

predictor is the default, and provides good performance without requiring user intervention to select appropriate scale factors.

- The `EXTERNAL` predictor is experimental and should not generally be used. This option is discussed in more detail in the in development manual.

4.9.1.2 Scale Factor

```
SCALE FACTOR = <real>scale_factor(1.0)
               [<real>first_scale_factor]
               [DURING <string list>period_names]
```

The `SCALE FACTOR` command controls the behavior of the `SCALE_FACTOR` predictor type. The velocity field of the previous step, multiplied by the value of `scale_factor`, is used to extrapolate from the previous solution to obtain a trial solution. The default value of 1.0 provides good performance on models that experience smooth loading.

Setting the value of `scale_factor` to 0.0 effectively turns off the predictor. Note, if an analysis has trouble reaching convergence turning off the predictor with a zero scale factor may be helpful. This is particularly true of analysis with contact or other potentially time discontinuous boundary conditions. Extrapolating the solution between steps with different boundary conditions may overshoot the solution in a way from which is difficult to recover.

The optional `first_scale_factor` parameter is used as the scale factor for the first step of a time period. If `first_scale_factor` is omitted, the default behavior is to use `scale_factor` for all time steps. Models are often subjected to loading that is mostly monotonic, but with discontinuities at the beginning of a new period. Using 1.0 and 0.0 for the values of `scale_factor` and `first_scale_factor`, respectively, often gives good performance in such cases.

The `SCALE FACTOR` command line can be appended with the `DURING` specification, as discussed in Section 4.2.1. This allows for the scale factor to be specified per time period.

4.9.1.3 Slip Scale Factor

```
SLIP SCALE FACTOR = <real>slip_factor(1.0)
                   [DURING <string list>period_names]
```

The `SLIP SCALE FACTOR` only has an effect in analyses with contact that use the kinematic contact enforcement option, see Section 8.9.3 for details.

The prediction of slip on slave nodes that are currently active in sliding contact can be controlled separately from other nodes in the model by using the `SLIP SCALE FACTOR` command line. The parameter `slip_factor` controls the scaling of the predicted slip on these nodes. If `slip_factor` is left at its default value of 1.0, contact slip will be predicted using the same scale factor used for the rest of the model. If the value of `slip_factor` is set to 0.0, the predicted solution will not allow any sliding contact nodes to slip. This command line applies to both the `SCALE_FACTOR` and the `SECANT` types of predictors.

The `SLIP SCALE FACTOR` command line can be appended with the `DURING` specification, as discussed in Section 4.2.1.

4.9.2 Level Predictor

```
LEVEL 1 PREDICTOR = <string>NONE|DEFAULT(DEFAULT)
```

The `LEVEL 1 PREDICTOR` command line controls the level 1 predictor. This command line appears in the `SOLVER` command block and is only applicable if the multilevel solver is used. The level 1 predictor estimates the solution for the next model problem based on the solution of the previous model problem. The default behavior is to enable this predictor selectively based on what solver controls are used (indicated by `DEFAULT`). The predictor can be turned off in all cases with the `NONE`.

4.9.3 Loadstep Predictor Input Example

The below example includes use of the predictor. The example predictor definition would be appropriate for an analysis with discontinuous loading during time period `p1` followed by a smoother loading during time period `p2`. Turning off the predictor in time period `p1` prevents overshooting solutions during sharp changes in loading. Turning back on the predictor in time period `p2` allows acceleration of solution in the more uniformly loaded time period.

```
begin solver
  begin loadstep predictor
    type = secant
    scale factor = 0 0 during p1
    scale factor = 1 0 during p2
  end
  Begin cg
    target relative residual = 1.0e-09
    Maximum Iterations      = 100
  end
end
```

4.10 Implicit Time Step Control

Time stepping in Sierra/SM is controlled by using a `TIME CONTROL` command block in the procedure scope. This command block controls the start time, the termination time, and the method by which time is advanced during the analysis. The analysis time can optionally be subdivided into a number of time periods. If the analysis is from time 0 to time T and it is split into three periods, the first period is defined from time 0 to time t_1 , the second period is defined from time t_1 to time t_2 , and the third period is defined from time t_2 to time T . (The times t_1 and t_2 are set in the input deck.) The sum of the times for the three periods is T .

There are many reasons for splitting an analysis into multiple time periods. Boundary conditions may be turned on or off in different time periods. Different solver parameters may also be used in different time periods.

Time stepping can be uniform during a solution period, or it can vary during a solution period by using a function to control the time step as a function of analysis time. With implicit time stepping an automatic time stepping scheme can also be used to automatically take larger time steps at points in the analysis when the solution is relatively easy and take smaller time steps when it becomes more difficult to solve a time step.

A description of the `TIME CONTROL` command block follows in Section 4.10.1. Section 4.10.2 describes the `ADAPTIVE TIME STEPPING` command block, which allows adaptation of the size of the time step based on solution difficulty. A simple example of a `TIME CONTROL` command block is presented in Section 4.10.3.

4.10.1 Command Blocks for Time Control and Time Stepping

```
BEGIN TIME CONTROL
  BEGIN TIME STEPPING BLOCK <string>time_block_name
    START TIME = <real>start_time_value
    BEGIN PARAMETERS FOR ADAGIO REGION <string>region_name
      #
      # Time control parameters specific to ADAGIO
      # are set in this command block.
      #
    END [PARAMETERS FOR ADAGIO REGION <string>region_name]
  END [TIME STEPPING BLOCK <string>time_block_name]
  TERMINATION TIME = <real>termination_time
END [TIME CONTROL]
```

Sierra/SM time control resides in a `TIME CONTROL` command block. The command block begins with the input line

```
BEGIN TIME CONTROL
```

and terminates an input line of the following form:

```
END [TIME CONTROL]
```

Time control is defined at the procedure scope. If a procedure has multiple regions the time control for each region is defined in a single procedure time control block.

An arbitrary number of `TIME STEPPING BLOCK` command blocks can be present to define individual time stepping periods within the `TIME CONTROL` command block. Each `TIME STEPPING BLOCK` command block contains the time at which the time period starts and a number of time steps for time stepping within the period. Each time period terminates at the start time of the following time period. The start times for the `TIME STEPPING BLOCK` command blocks must appear in increasing time. The example in Section 4.10.3 shows the overall structure of the `TIME CONTROL` command block.

In the above input lines, the parameters are as follows:

- The string `time_block_name` is a name for the time period. Every time period must have a unique name. These names are referenced to control solution parameters or to activate/deactivate functionality.
- The real value `start_time_value` is the start time for this `TIME STEPPING BLOCK` time period. A time period goes from its start time until the start time of the next period or the termination time. The start time may be negative.
- The string `region_name` is the name of the region affected by the parameters (see Section 2.2).

The final termination time for the analysis is given by `TERMINATION TIME` command. The input `termination_time` is the time at which the analysis will stop. The `TERMINATION TIME` command line appears inside the `TIME CONTROL` command block but outside of any `TIME STEPPING BLOCK` command block.

The `TERMINATION TIME` command line can appear before the first `TIME STEPPING BLOCK` command block or after the last `TIME STEPPING BLOCK` command block. It is permissible to have `TIME STEPPING BLOCK` command blocks with start times greater than the termination time; but these time periods will not be executed and will have no effect on the analysis. Only one `TERMINATION TIME` command line can appear in the `TIME CONTROL` command block. If more than one of these command lines appears, an error is generated.

Nested inside the `TIME STEPPING BLOCK` command block is a `PARAMETERS FOR ADAGIO REGION` command block containing parameters that control the time stepping.

```
BEGIN PARAMETERS FOR ADAGIO REGION <string>region_name
    TIME INCREMENT = <real>time_increment_value
    NUMBER OF TIME STEPS = <integer>nsteps
    TIME INCREMENT FUNCTION = <string>time_function
END [PARAMETERS FOR ADAGIO REGION <string>region_name]
```

The parameters in the `BEGIN PARAMETERS FOR ADAGIO REGION` block are specific to an implicit analysis. Note, similar parameters in a `BEGIN PARAMETERS FOR PRESTO REGION` block are specific to an explicit analysis 3.

The command block begins with an input line of the form

```
BEGIN PARAMETERS FOR ADAGIO REGION <string>region_name
```

and is terminated with an input line of the following form:

```
END [PARAMETERS FOR ADAGIO REGION <string>region_name]
```

As noted previously, the string `region_name` is the name of the region to which the parameters apply. The command lines nested inside the `PARAMETERS FOR ADAGIO REGION` command block are used to control the way time stepping occurs, and are described in Sections 4.10.1.1 through 4.10.1.3. Only one of these command lines (`TIME INCREMENT`, `NUMBER OF TIME STEPS`, or `TIME INCREMENT FUNCTION`) may be used in a given time period.

4.10.1.1 Time Increment

```
TIME INCREMENT = <real>time_increment_value
```

The `TIME INCREMENT` command line directly specifies the size of a uniform time step used during a time period. The size of the time step is specified with the parameter `time_increment_value`.

4.10.1.2 Number of Time Steps

```
NUMBER OF TIME STEPS = <integer>nsteps
```

The `NUMBER OF TIME STEPS` command line specifies the number of uniform time steps used during a time period. The number of time steps is specified with the parameter `nsteps`. If this command line is used, the time step size is computed by dividing the length of the time period by the specified number of time steps.

4.10.1.3 Time Increment Function

```
TIME INCREMENT FUNCTION = <string>time_function
```

The `TIME INCREMENT FUNCTION` command line allows the time step to be specified as a function of analysis time. The function used to control the time step is referenced by the parameter `time_function`. The actual function is defined within a `FUNCTION` command block within the `SIERRA` scope. It is often convenient to use a function of the `PIECEWISE CONSTANT` type (see Section 2.1.5) with this command line. Using a function to control time incrementation allows for the time increment to be conveniently switched many times during an analysis without creating a new time block every time the time step size needs to be changed.

4.10.2 Adaptive Time Stepping

```
BEGIN ADAPTIVE TIME STEPPING
  METHOD = <string>SOLVER|SOLVER_AVERAGE|MATERIAL|USER(SOLVER)
    [DURING <string list>period_names]
  TARGET ITERATIONS = <integer>target_iter(0)
    [DURING <string list>period_names]
  ITERATION WINDOW = <integer>iter_window(0)
    [DURING <string list>period_names]
  CUTBACK FACTOR = <real>cutback_factor(0.5)
    [DURING <string list>period_names]
  GROWTH FACTOR = <real>growth_factor(1.5)
    [DURING <string list>period_names]
```

```

MAXIMUM FAILURE CUTBACKS = <integer>max_cutbacks(5)
    [DURING <string list>period_names]
MAXIMUM MULTIPLIER = <real>max_multiplier(infinity)
    [DURING <string list>period_names]
MINIMUM MULTIPLIER = <real>min_multiplier(1.0e-8)
    [DURING <string list>period_names]
RESET AT NEW PERIOD = TRUE|FALSE(TRUE)
    [DURING <string list>period_names]
ACTIVE PERIODS = <string list>period_names
USER CRITERIA GLOBAL <string>var_name <|<=|=|>=> <real>cmp_val
END [ADAPTIVE TIME STEPPING]

```

An Adagio region has the capability to adapt the time step size during an analysis based on either solution difficulty or on feedback from material models. This capability is enabled by inserting an `ADAPTIVE TIME STEPPING` command block in the region scope. If this adaptive command block is not present in the Adagio region, the time stepping specified in the `TIME CONTROL` command block will be used exclusively. In addition to adjusting the size of time steps as the analysis proceeds the adaptive time stepping algorithm can attempt to solve a load step by using a smaller time step if a solution fails to converge. Only a single adaptive stepping time block can be used in a region.

The adaptive time stepping algorithm works by computing a multiplier that is applied to the time step specified in the `TIME CONTROL` command block. Thus, even if adaptive time stepping is used the base time step must be specified for all loading periods in the `TIME CONTROL` command block. For the initial load step in each solution period the Adagio region always uses the prescribed time step. In subsequent steps or after convergence failure the adaptive time stepping algorithm computes a factor that is multiplied by the prescribed time step to adjust the time step.

The command lines nested within the `ADAPTIVE TIME STEPPING` command are used to control the behavior of this feature and are described in Sections [4.10.2.1](#) through [4.10.2.10](#).

Many of these command lines can optionally be set for a specific time period. To set a parameter for a specific time period the `DURING` keyword followed by a list of applicable period names to a command line may be used (see Section [4.2.1](#)). Multiple instances of these command lines can exist to set the parameter differently for different time periods. If a command line is not appended with the `DURING` specification, it is used to set the default behavior for all time periods. Setting a default in this way does not preclude setting a period-specific value for that parameter with another instance of the same command line.

4.10.2.1 Method

```

METHOD = <string>SOLVER|SOLVER_AVERAGE|MATERIAL|USER(SOLVER)
    [DURING <string list>period_names]

```

The `METHOD` command line is used to select the adaptive time stepping algorithm. The default `SOLVER` method adapts the time step based on the difficulty of the solution over a time step. Solution difficulty is determined based on the number of iterations required to achieve convergence

during a time step. If the `SOLVER` method is used, the `TARGET ITERATIONS` command line can be used to specify the target iterations used by this method to adapt the time step.

The `SOLVER_AVERAGE` method can be used when the problem has a multi level solver control such as control contact. With the `SOLVER_AVERAGE` option the difficulty is determined as the average number of iterations the solver takes per model problem instead of the accumulated number of iterations taken per time step.

The `MATERIAL` method uses the material models to inform an appropriate step size. A material may provide a maximum time step to prevent too much nonlinearity over a step. Note, only a handful of materials will define time steps and this option is primarily relevant to materials that have time dependent creep or other rate effects. The material suggested time step multiplier will still be bounded by the adaptive time stepping block input parameters for `MAXIMUM MULTIPLIER`, `MINIMUM MULTIPLIER`, `GROWTH FACTOR`, and `CUTBACK FACTOR`.

The `USER` method must be paired with a `USER CRITERIA GLOBAL`, see Section 4.10.2.11. When the criteria is satisfied a time step cutback is forced using the `CUTBACK FACTOR`, a convergence failure is mimicked and the current solution is thrown away. The time step is still bounded by the adaptive time stepping block input parameters but the `GROWTH FACTOR` is unused.



Warning: The `USER` method does not currently support the optional `DURING`.

4.10.2.2 Target Iterations

```
TARGET ITERATIONS = <integer>target_iter(0)
[DURING <string list>period_names]
```

The solver method for adaptive time stepping adjusts the time step to achieve a target level of difficulty. The level of difficulty is determined by the total number of core solver iterations required to solve for a time step. The `TARGET ITERATIONS` command is used to specify a target number of iterations per time step. This command line is optional.

The specified value of `target_iter` is used in conjunction with the value of `iter_window` (specified with the `ITERATION WINDOW` command documented in Section 4.10.2.3) to control the next step size. If the number of iterations for the previous step is greater than (`target_iter - iter_window`), and less than (`target_iter + iter_window`), the step size is kept the same. If the number of iterations is greater than (`target_iter + iter_window`), the step size is decreased, and if it is less than (`target_iter - iter_window`), the step size is increased.

If the target iterations is undefined or zero solver will not adjust the time step based on iteration count. The solver will cutback the time step if an element inverts, the solver fails to converge, or a material model suggests a smaller time step.

4.10.2.3 Iteration Window

```
ITERATION WINDOW = <integer>iter_window
```

```
[DURING <string list>period_names]
```

The `ITERATION WINDOW` command is used to specify the size of the window for the iteration count used in the solver type of adaptive time stepping. See Section 4.10.2.2 for an explanation of how this is used in conjunction with the specified target iterations to adaptively choose the time step size. The default value of `iter_window` is the value specified in the `TARGET ITERATIONS` command line divided by 5.

4.10.2.4 Cutback Factor

```
CUTBACK FACTOR = <real>cutback_factor(0.5)
[DURING <string list>period_names]
```

The `CUTBACK FACTOR` command line controls the amount by which the next time step is reduced if the solution is difficult. The parameter `cutback_factor` specifies the multiplier that is applied to the current time step with each cutback. The default value for the `CUTBACK FACTOR` is 0.5, the factor must be between zero and one. The cutback factor parameter also controls the amount that the current time step is cut back if a solution fails. The `CUTBACK FACTOR` command line can be appended with the `DURING` specification, as discussed in Section 4.2.1.

4.10.2.5 Growth Factor

```
GROWTH FACTOR = <real>growth_factor(1.5)
[DURING <string list>period_names]
```

The `GROWTH FACTOR` command line controls the amount by which the next time step is increased if the solution is easy. The parameter `growth_factor`, which has a default value of 1.5, specifies the multiplier (a 50 percent increase) that is applied to the current time step each time the time step is increased. The `GROWTH FACTOR` command line can be appended with the `DURING` specification, as discussed in Section 4.2.1.

4.10.2.6 Maximum Failure Cutbacks

```
MAXIMUM FAILURE CUTBACKS = <integer>max_cutbacks(5)
[DURING <string list>period_names]
```

The `MAXIMUM FAILURE CUTBACKS` command line controls how many cutbacks to a time step can be taken if a solution fails to converge. The parameter `max_cutbacks` has a default value of 5 if adaptive time stepping is enabled. If an `ADAPTIVE TIME STEPPING` command block is not present, the code will exit rather than attempting to cut back after an unsuccessful solution. The `MAXIMUM FAILURE CUTBACKS` command line can be appended with the `DURING` specification, as discussed in Section 4.2.1.

4.10.2.7 Maximum Multiplier

```
MAXIMUM MULTIPLIER = <real>max_multiplier(infinity)
[DURING <string list>period_names]
```


Time steps are adaptively adjusted by updating a multiplier that is applied to the base time step specified in the `TIME CONTROL` command block. By default, there are no limits on the size of this multiplier. The `MAXIMUM MULTIPLIER` command line can be used to limit the growth of the time step. The parameter `max_multiplier` specifies the upper bound for the multiplier. The largest possible time step in an analysis is the product of `max_multiplier` and the baseline time step size. This baseline size is specified in the `TIME CONTROL` command block. The `MAXIMUM MULTIPLIER` command line can be appended with the `DURING` specification, as discussed in Section 4.2.1.

4.10.2.8 Minimum Multiplier

```
MINIMUM MULTIPLIER = <real>min_multiplier(1.0e-8)
[DURING <string list>period_names]
```

Time steps are adaptively adjusted by updating a multiplier that is applied to the base time step specified in the `TIME CONTROL` command block. By default, the `MINIMUM MULTIPLIER` is set to $1.0e-8$. The `MINIMUM MULTIPLIER` command line can be used to limit the shrinkage of the time step. The parameter `min_multiplier` specifies the lower bound for the multiplier. The smallest possible time step in an analysis is the product of `min_multiplier` and the baseline time step size. The `MINIMUM MULTIPLIER` command line can be appended with the `DURING` specification, as discussed in Section 4.2.1.

4.10.2.9 Reset at New Period

```
RESET AT NEW PERIOD = TRUE|FALSE(TRUE)
[DURING <string list>period_names]
```

This command controls whether the time step is reset to the user-specified base time step at the beginning of a new time period. If this parameter is set to `TRUE`, which is the default value, the time step is reset. If this parameter is set to `FALSE`, the adaptive time stepping is maintained from the previous period. If the base time step is changed in the new period, the multiplier is scaled to maintain the time step originally calculated for the next step. The multiplier is scaled by the ratio: time increment of the previous period/time increment of the new period. If the resulting multiplier is outside of the specified range, it will be adjusted to stay within that range and the time step will change. In summary, the first time step of the new period will be equal to the base time step of the previous period times the scaled multiplier. The value of this parameter can optionally be specified per time period with the `DURING` option (see Section 4.2.1). If adaptive time stepping is inactive for a given period, the time step will always be reset even if it is requested that the adaptive time step not be reset.

4.10.2.10 Active Periods

```
ACTIVE PERIODS = <string list>period_names
```

If the `ADAPTIVE TIME STEPPING` command block is present in the Adagio region, it is active for all periods by default. Adaptive time stepping can be activated for specific time periods with

the `ACTIVE PERIODS` command line. See Section 2.6 for more information about this optional command line.

4.10.2.11 User Criteria Global

```
USER CRITERIA GLOBAL <string>var_name <|<=|=|>=|> <real>cmp_val
```

The `USER CRITERIA GLOBAL` command is for paring with method `USER`. At the end of a solve, the global variable `var_name` is compared to the `cmp_val` with the specified comparison type. If the comparison is true a cutback to the timestep is forced. The `CUTBACK FACTOR` is used to reduce the time step while respecting bounds on the timestep defined by `MINIMUM MULTIPLIER` and `MAXIMUM FAILURE CUTBACKS`.

4.10.3 Time Control Example

The following is a simple example of a `TIME CONTROL` command block:

```
BEGIN TIME CONTROL
  BEGIN TIME STEPPING BLOCK p1
    START TIME = 0.0
    BEGIN PARAMETERS FOR ADAGIO REGION adagio_region
      NUMBER OF TIME STEPS = 10
    END
  END
  BEGIN TIME STEPPING BLOCK p2
    START TIME = 1.0
    BEGIN PARAMETERS FOR ADAGIO REGION adagio_region
      TIME INCREMENT = 0.2
    END
  END
  TERMINATION TIME = 2.0
END
```

The first `TIME STEPPING BLOCK`, `p1`, begins at time 0.0, the initial start time, and terminates at time 1.0. The first time period will take ten time steps each of time 0.1 to reach time 1.0.

The second `TIME STEPPING BLOCK`, `p2`, begins at time 1.0 and terminates at time 2.0, the time listed on the `TERMINATION TIME` command line. The second time period will take five time steps each of time 0.2 to reach the final end time of 2.0.

The `TIME STEPPING BLOCK` names `p1` and `p2` can be referenced elsewhere to activate boundary conditions or control the solver in different ways during the two analysis time periods.

4.11 Implicit Dynamic Time Integration

Sierra/SM has the ability to perform implicit solution of both quasistatic and dynamic problems. In quasistatic calculations, the solution for static equilibrium is obtained at each step, ignoring the inertial mass and acceleration terms of the equations of motion. In dynamic problems, the inertial terms are included.

4.11.1 Implicit Dynamics

```
BEGIN IMPLICIT DYNAMICS
  ACTIVE PERIODS = <string list>period_names
  USE HHT INTEGRATION
  ALPHA = <real>alpha(-0.1) [DURING <string list>period_names]
  BETA = <real>beta(0.3025) [DURING <string list>period_names]
  GAMMA = <real>gamma(0.6) [DURING <string list>period_names]
  CONTACT TIMESTEP = ON|OFF(ON)
    [DURING <string list>period_names]
END [IMPLICIT DYNAMICS]
```

The `IMPLICIT DYNAMICS` command block enables implicit dynamics and contains commands to control the behavior of implicit time integration. This command block is placed in the `ADAGIO REGION` command block. Formerly, it was necessary to create a new procedure for switching between implicit dynamics and quasistatics in an analysis. Now, these different analysis types can all be done within a single region, and the command lines in the `IMPLICIT DYNAMICS` command block can be used to enable or disable implicit dynamics for specific time periods within a single region. Sections 4.11.1.1 through 4.11.1.4 describe the command lines in this command block.

4.11.1.1 Active Periods

```
ACTIVE PERIODS = <string list>period_names
```

If the `IMPLICIT DYNAMICS` command block is present in the Adagio region, it is active for all periods by default. Implicit dynamics can be activated for specific time periods with the `ACTIVE PERIODS` command line. See Section 2.6 for more information about this optional command line.

4.11.1.2 Use HHT Integration

```
USE HHT INTEGRATION
```

Adagio currently only supports the HHT algorithm for implicit time integration. The `USE HHT INTEGRATION` command line is provided as a placeholder to allow for other time integrators in the future, but currently this command line has no effect.

4.11.1.3 HHT Parameters

```
ALPHA = <real>alpha(-0.1) [DURING <string list>period_names]
BETA = <real>beta(0.3025) [DURING <string list>period_names]
GAMMA = <real>gamma(0.6) [DURING <string list>period_names]
```

The `ALPHA`, `BETA`, and `GAMMA` command lines specify the HHT integration parameters. These parameters can all be specified for the entire analysis, or they can vary by solution period by appending them with the optional `DURING` specification (see Section 4.2.1).

- The `ALPHA` command line specifies α , which is the dissipation factor applied to the internal force vector. The value of `alpha` controls numerical damping and must always be less than or equal to zero. Its default value of -0.1 results in slight numerical damping of modes with frequencies higher than can be resolved with a given time step. To maintain second-order accuracy, the value of `alpha` should not be less than $-\frac{1}{3}$.
- The `BETA` command line specifies the stability parameter β for time integrators in the Newmark family. The default value of `beta` is 0.3025. It should have a value of $0.25(1 - \alpha)^2$ to maintain second-order accuracy.
- The `GAMMA` command line specifies the dissipation factor γ for Newmark time integrators. The default value of `gamma` is 0.6. It should have a value of $0.5 - \alpha$ to maintain second-order accuracy.

The HHT time integrator [5] is used to integrate the semi-discrete equations of motion in time. The behavior of the HHT integrator is controlled with three parameters: α , β , and γ . With proper selection of these parameters, the HHT integrator is unconditionally stable. If $\alpha = 0$, the HHT integrator reduces to the Newmark method [6], for which unconditional stability requires $\beta \geq (\gamma + 1/2)^2$. The trapezoidal rule is recovered if $\alpha = 0$, $\beta = 0.25$, and $\gamma = 0.5$. For a detailed discussion of the theory of implicit time integrators, see [7].

4.11.1.4 Implicit Dynamic Timestep Cutback for Contact

```
CONTACT TIMESTEP = ON|OFF (ON)
[DURING <string list>period_names]
```

When running an implicit dynamics simulation, errors can sometimes occur when too large of a time step is taken for solving a problem involving contact. For example blocks move completely through each other during a single time step contact may be missed or solved inaccurately. The `CONTACT TIMESTEP` command line specifies whether the time step can be cut back for better integration of contact impact. The default is `ON` which allows for the time step to be cut. This parameter can also vary by period by appending the command lines with the optional `DURING` specification (see Section 4.2.1).

4.12 Deprecated Commands

The command `NUMBER OF SMOOTHING ITERATIONS` in the full tangent preconditioner block is deprecated. The command `MINIMUM SMOOTHING ITERATIONS` is a direct replacement and should be used instead.

References

- [1] M.L. Blanford, M.W. Heinstein, and S.W. Key. JAS3D – a multi-strategy iterative code for solid mechanics analysis users’ instructions, release 2.0, September 2001.

- [2] R. Fletcher and C.M. Reeves. Function minimization by conjugate gradients. *The Computer Journal*, 7:149–153, 1964.
- [3] C. Farhat, M. Lesoinne, and K. Pierson. A scalable dual-primal domain decomposition method. *Numerical Linear Algebra with Applications*, 7:687–714, 2000.
- [4] C. Farhat, M. Lesoinne, P. LeTallec, K. Pierson, and D. Rixen. FETI-DP: A dual-primal unified FETI method – part i: A faster alternative to the two-level FETI method. *International Journal for Numerical Methods in Engineering*, 50:1523–1544, 2001.
- [5] H.M. Hilber, T.J.R. Hughes, and R.L. Talor. Improved numerical dissipation for time integration algorithms in structural dynamics. *Earthquake Engineering and Structural Dynamics*, 5: 283–292, 1977.
- [6] N.M. Newmark. A method of computation for structural dynamics. *Journal of Engineering Mechanics, ASCE*, 85(EM3):67–94, 1959.
- [7] T.J.R. Hughes. *The Finite Element Method—Linear Static and Dynamic Finite Element Analysis*. Prentice-Hall, Inc, Englewood Cliffs, NJ, 1987.

Chapter 5

Material Models

This chapter describes material models that can be used in conjunction with the elements in Sierra/SM. Most of the material models can be used in either implicit or explicit analyses. Even though a material model can be used by both implicit and explicit, usage of the model may be better suited for the type of problems solved implicit verses explicit. For example, a material model that was built to characterize behavior over a long time would be better suited for use in Adagio/Implicit than in Presto/Explicit.

The material models described in this chapter are, in general, applicable to solid elements. Structural elements, such as shells and beams, have a more limited set of material models. Chapter 6 describes the element library, including which material models are available for the various elements. The introduction to Chapter 6 summarizes all the element types available in Sierra/SM. For each element type, a list of available material models is provided.

State variable output is available for many materials. See Section 9.10.2 to learn how to output the state variables for materials or to the specific material model sections for the list of computed state variables for that material model.

Most material models used by Sierra/SM are defined by the LAME material library [1].

General Model Form. MATERIAL command blocks appear in the SIERRA scope in the general form shown below.

```
BEGIN MATERIAL <string>mat_name
#
# Command lines and command blocks for material
# models appear in this scope.
#
END [MATERIAL <string>mat_name]
```

MATERIAL command blocks are physics independent in the sense that the information in them can be shared by more than one application. The command block begins with the line:

```
BEGIN MATERIAL <string>mat_name
```

and terminates with the line:

```
END [MATERIAL <string>mat_name]
```

Here, the string `mat_name` is a user-specified name for the command block. This name is typically descriptive of the material being modeled, e.g., `aluminum_t6061`.

Within a `MATERIAL` command block, there will usually be at least one `BEGIN PARAMETERS FOR MODEL` command block as well as other general material command lines used to describe particular material models. The general material command lines, if present, are listed first, followed by one or more material-model command blocks. The general material command lines may be used to specify the density of the material, the Biot's coefficient, and the application of temperatures and thermal strains. Each material-model command block follows the naming convention of `PARAMETERS FOR MODEL model_name`, where `model_name` identifies a particular material model, such as elastic, elastic-plastic, or orthotropic crush. Each such command block contains all the parameters needed to describe a particular material model.

As noted above, more than one material-model command block can appear within a `MATERIAL` command block. For example, a `MATERIAL` command block called `steel` could have two nested material-model command blocks. One of the material-model command blocks would provide an elastic model for `steel` and the other an elastic-plastic model for `steel`. The general form of a `MATERIAL` command block would be as follows:

```
BEGIN MATERIAL <string>mat_name
#
# General material command lines
DENSITY = <real>density_value
BIOTS COEFFICIENT = <real>biots_coefficient_value
#
THERMAL {LOG|ENGINEERING} STRAIN FUNCTION =
    <string>thermal_strain_function
# or all three of the following
THERMAL {LOG|ENGINEERING} STRAIN X FUNCTION =
    <string>thermal_strain_x_function
THERMAL {LOG|ENGINEERING} STRAIN Y FUNCTION =
    <string>thermal_strain_y_function
THERMAL {LOG|ENGINEERING} STRAIN Z FUNCTION =
    <string>thermal_strain_z_function
THERMAL {LOG|ENGINEERING} STRAIN DIRECTION
    <string>dir_name FUNCTION = <string>funcName
#
ARTIFICIAL ENGINEERING STRAIN FUNCTION =
    <string>artificial_strain_function
# or all three of the following
ARTIFICIAL ENGINEERING STRAIN X FUNCTION =
    <string>artificial_strain_x_function
ARTIFICIAL ENGINEERING STRAIN Y FUNCTION =
    <string>artificial_strain_y_function
ARTIFICIAL ENGINEERING STRAIN Z FUNCTION =
```



```

    <string>artificial_strain_z_function
ARTIFICIAL ENGINEERING STRAIN DIRECTION
    <string>dir_name FUNCTION = <string>funcName

BEGIN PARAMETERS FOR MODEL <string>model_name1
    #
    # Parameters for material model model_name1
END PARAMETERS FOR MODEL <string>model_name1
#
BEGIN PARAMETERS FOR MODEL <string> model_name2
    #
    # Parameters for material model model_name2
END PARAMETERS FOR MODEL <string> model_name2
#
# Additional model command blocks if required
#
END [MATERIAL <string>mat_name]

```

In the above general form for a MATERIAL command block, the string `model_name1` could be ELASTIC and the string `model_name2` could be ORTHOTROPIC CRUSH. Typically, however, only one material model would be desired for a given block, and the MATERIAL command block would have only one PARAMETERS FOR MODEL command block. A particular material model may only appear once within a given MATERIAL command block.

Although multiple material models can be defined for one material within a MATERIAL command block, only one material model is actually used for a given element block during an analysis. The ability to define multiple constitutive models for one material is provided as a convenience to enable easily switching between models. The material name and the model name are both referenced when material models are assigned to element blocks within the FINITE ELEMENT MODEL command block, which is described in Section 6.1.

This chapter is organized according to the general form for the MATERIAL command block. Section 5.1 discusses the DENSITY command line, the BIOTS COEFFICIENT command line, and the command lines used for thermal strains, and also explains how temperatures and thermal strains are applied. Section 5.2 describes each of the material models that are used by Sierra/SM. References applicable to Sierra/SM are listed at the end of the chapter.

As indicated in the introductory material, not all the material models are applicable to all element types. As one example, there are the material models available for use with cohesive zone elements or with Dash contact, as detailed in Section 5.3. As another example, there is a one-dimensional elastic material model that is used for a truss element but is not applicable to solid elements such as hexahedra or tetrahedra. For this particular example, the specific material-model usage is hidden from the user. If the user specifies a linear elastic material model for a truss, the one-dimensional elastic material model is used. If the user specifies a linear elastic material model for a hexahedron, a full three-dimensional elastic material model is used. As another example, the energy-dependent material models in Presto can only be used for solid elements such as hexahedra and tetrahedra. (Chapter 6 indicates what material models are available for which element formulations.)

Nonlinear material models have state variables. See Section [9.10.2](#) for more information on how to output the state variables for the nonlinear material models.

Brief descriptions of the material models are presented in this chapter. More detailed descriptions can be found in the references for most material models.

5.1 General Material Commands

A `MATERIAL` command block may include command lines applicable to all the material models specified within the command block. These command lines related to density, to Biot's coefficient, and the thermal strain behavior are discussed, respectively, in Section 5.1.1, Section 5.1.2, and Section 5.1.4. Additionally several material models share similar behavior for certain commands such as elastic constants and the beta parameter discussed in Section 5.1.5 and Section 5.1.6.

5.1.1 Density Command

```
DENSITY = <real>density_value
```

This command line specifies the density of the material. The units of the input parameter `density_value` are mass per unit volume.

A `MATERIAL` command block can contain one or more `PARAMETERS FOR MODEL` command blocks. The specified `density_value` will be used with all models described in these `PARAMETERS FOR MODEL` command blocks.

5.1.2 Biot's Coefficient Command

```
BIOTS COEFFICIENT = <real>biots_value
```

This command line specifies the Biot's coefficient of the material. Biot's coefficient is used with the pore pressure capability (commonly called an effective stress model). See Section 7.11 for more information on pore pressure. If not given, the value defaults to 1.0 which is a common value for many rock properties. This parameter is unitless. The pore pressure capability is implemented as the following effective stress model

$$\sigma_{effective} = \sigma - \alpha P \quad (5.1)$$

where P is the pressure and α is Biot's coefficient.

A `MATERIAL` command block can contain one or more `PARAMETERS FOR MODEL` command blocks. The specified `biots_value` will be used with all models described in these `PARAMETERS FOR MODEL` command blocks.

5.1.3 General Artificial Strains

Sierra/SM allows a user to apply an artificial strain to a material. Artificial strains may be applied via the Artificial Strain commands or via the Thermal Strain commands (5.1.4). General artificial strains may be useful for fitting parts together for a preload analysis or modeling a variety of other physics that could cause material expansion or contraction (such as radiation induced expansion, material desiccation, etc.)

Artificial strains may also be defined via a boundary condition. See Section 7.14.7.

The artificial strain commands are placed in the `MATERIAL` command block.

```

ARTIFICIAL ENGINEERING STRAIN FUNCTION =
  <string>artificial_strain_function
#
# Or all three of the following
#
ARTIFICIAL ENGINEERING STRAIN X FUNCTION =
  <string>artificial_strain_x_function
ARTIFICIAL ENGINEERING STRAIN Y FUNCTION =
  <string>artificial_strain_y_function
ARTIFICIAL ENGINEERING STRAIN Z FUNCTION =
  <string>artificial_strain_z_function
#
# Or one to three of the following
#
ARTIFICIAL ENGINEERING STRAIN DIRECTION
  <string>dir_name FUNCTION = <string>funcName

```

The `ARTIFICIAL ENGINEERING STRAIN FUNCTION` defines an isotropic strain.

The X, Y, and Z functions may define anisotropic strain. Note, the X, Y, and Z directions correspond to the material coordinate system. For most solid element materials, the material coordinate system is aligned with the global X, Y, and Z axes. For orthotropic solid materials, the X, Y, and Z thermal strains are applied in the material orthotropic R, S, and T coordinate system (which may vary element to element). For two dimensional shell and membrane elements, the X and Y directions are in the plane of the shell, and Z is the through thickness direction. Z strains generally have no effect on shells. See Section 6.2.4 for a discussion of the shell *rst* coordinate system. For one dimensional beam or truss elements, the X direction is along the length of the beam, and the Y and Z lie in the orthogonal directions. Y and Z strains generally have no effect on beams or trusses.

The `STRAIN DIRECTION` function defines strain in an arbitrary direction (note, still in the material coordinate system). Up to three orthogonal direction functions may be used.

All of the artificial strain functions define the material engineering strain with respect to time. If a material is using artificial strain, it must define either one isotropic function or all three anisotropic functions.

It is highly important to note that there is zero artificial strain applied at the reference (original) model configuration. Thus artificial strains are calculated as the difference in strain function from the current time back to the reference time (analysis start time). For clarity, it is recommended that the artificial strain function be zero at the start time of the analysis. Artificial strain is a component of Non-Material strain; consequently, it is a strain that does not produce stress from the material response. The output variable names for artificial strain can be found in Table 9.10, Table 9.11, Table 9.12, Table 9.13, or Table 9.15 depending on the element type it is applied to.

5.1.4 Thermal Strain Behavior

Isotropic and orthotropic thermal strains may be defined for use by material models listed in a `MATERIAL` command block. Section 5.1.4.1 describes the command lines that are used to define

the thermal strain behavior. These command lines must be used in conjunction with other command blocks outside of a `MATERIAL` command block for the calculations of thermal strains to be activated. Section 5.1.4.2 explains the process for activating thermal strains.

5.1.4.1 Defining Thermal Strains

```

THERMAL {LOG|ENGINEERING} STRAIN FUNCTION =
    <string>thermal_strain_function
#
# or all three of the following
#
THERMAL {LOG|ENGINEERING} STRAIN X FUNCTION =
    <string>thermal_strain_x_function
THERMAL {LOG|ENGINEERING} STRAIN Y FUNCTION =
    <string>thermal_strain_y_function
THERMAL {LOG|ENGINEERING} STRAIN Z FUNCTION =
    <string>thermal_strain_z_function
#
# Or one to three of the following
#
THERMAL {LOG|ENGINEERING} STRAIN DIRECTION
    <string>dir_name FUNCTION = <string>funcName

```

A `MATERIAL` command block may include command lines that define thermal strain behavior. It is possible to specify either an isotropic thermal-strain field using the command line `STRAIN FUNCTION` or an orthotropic thermal-strain field using the command lines `STRAIN X FUNCTION`, `STRAIN Y FUNCTION`, and `STRAIN Z FUNCTION`. For any of these command lines, the user supplies a thermal strain function (via a `FUNCTION` command block), which defines the thermal strain as a function of temperature. Thermal strain is a component of Non-Material strain; consequently, it is a strain that does not produce stress from the material response. The computed thermal strain is then subtracted from the strain passed to the material model.

Thermal strains may be specified in either engineering or log strain values. Note, the older version of these commands `THERMAL STRAIN FUNCTION = specified log strain`. Thermal strain data is often provided in engineering strain units, though the native strain measure in side of Adagio/Presto is log strain. At small strain values, log strain and engineering strain are nearly identical.

The `THERMAL STRAIN FUNCTION` defines an isotropic strain.

The X, Y, and Z functions define anisotropic strain. Note, the X, Y, and Z directions correspond to the material coordinate system. For most solid element materials, the material coordinate system is aligned with the global X, Y, and Z axes. For orthotropic solid materials, the X, Y, and Z thermal strains are applied in the material orthotropic R, S, and T coordinate system (which may vary element to element). For two dimensional shell and membrane elements, the X and Y directions are in the plane of the shell, and Z is the through thickness direction. Z strains generally have no effect on shells. See Section 6.2.4 for a discussion of the shell *rst* coordinate system. For one dimensional beam or truss elements, the X direction is along the length of the beam, and the Y and Z lie in the orthogonal directions. Y and Z strains generally have no effect on beams or trusses.

The `STRAIN DIRECTION` function defines strain in an arbitrary direction (note, still in the material coordinate system). Up to three orthogonal direction functions may be used.

If an isotropic thermal strain field is to be applied, the `THERMAL {LOG|ENGINEERING} STRAIN FUNCTION` command line is placed in the `MATERIAL` command block outside of the specifications of any material models. Such placement is necessary because the isotropic thermal strain is a general material property, not a property that is specific to any particular constitutive model such as `ELASTIC` or `ELASTIC-PLASTIC`. The input value of `thermal_strain_function` is the name of the function that defines thermal strain as a function of temperature for the material model described in this particular `MATERIAL` command block. The function is defined within the `SIERRA` scope using a `FUNCTION` command block. For more information on how to set the input to compute thermal strains and how to apply temperatures, see Section 5.1.4.2.

The specification of an orthotropic thermal strain field requires that all three of the `THERMAL STRAIN X FUNCTION`, `THERMAL STRAIN Y FUNCTION`, and `THERMAL STRAIN Z FUNCTION` command lines be placed in the `MATERIAL` command block. All three command lines must be provided, even when there is no thermal strain in one or more directions. The values of `thermal_strain_x_function`, `thermal_strain_y_function`, and `thermal_strain_z_function` are the names of the functions for thermal strains in the *X*-direction, the *Y*-direction, and the *Z*-direction, respectively. These functions are defined within the `SIERRA` scope using `FUNCTION` command blocks. To specify that there should be no thermal strain in a given direction, use a function that always evaluates to zero for that direction.

If one the `THERMAL {LOG|ENGINEERING} STRAIN FUNCTION` command lines are used for one of the several of the material models, as discussed in Section 5.1.4.2. The model external artificial strain and the model internal artificial strain will be multiplied together. The use of a thermal strain is identified in the descriptions of the material models in Section 5.2 by the notation “`thermal strain option`”.

If both artificial and thermal strain are defined simultaneously the two strain components are additive.

The output variable names for thermal strain can be found in Table 9.9.

5.1.4.2 Activating Thermal Strains

Sierra/SM has the capability to compute thermal strains on three-dimensional continuum, two-dimensional (shell, membrane), and one-dimensional (beam, truss) elements. Three inputs are required to activate thermal strains:

- One or more thermal strain functions (strain as a function of temperature) must be defined. Each thermal strain function is defined with a `FUNCTION` command block. (This function is the standard function definition that appears in the `SIERRA` scope.) The thermal strain function gives the total thermal strain associated with a given temperature. It is the change in thermal strain with the change in temperature that gives rise to thermal stresses in a body.
- The material models used by blocks that experience thermal strain must have their thermal strain behavior defined. The command lines for defining isotropic and orthotropic thermal strain are described in Section 5.1.4.1. Materials with isotropic thermal strain use the `THERMAL {LOG|ENGINEERING} STRAIN FUNCTION` command line,

while those with orthotropic thermal strain must define thermal strain in all three directions using the `THERMAL {LOG|ENGINEERING} STRAIN X FUNCTION`, `THERMAL {LOG|ENGINEERING} STRAIN Y FUNCTION`, and `THERMAL {LOG|ENGINEERING} STRAIN Z FUNCTION` command lines. These inputs can be used with all material models with the exception of the following: elastic three-dimensional orthotropic, Mooney-Rivlin, NLVE three-dimensional orthotropic, Swanson, and viscoelastic Swanson. These models require their own model-specific inputs to define thermal strain and must not use these standard commands. Information for defining thermal strains is provided in the individual descriptions of these models in Section 5.2.

- A temperature field must be applied to the affected blocks. The boundary condition command block to specify the application of temperatures is `PRESCRIBED TEMPERATURE`. A description of the `PRESCRIBED TEMPERATURE` command block is given in Section 7.10.

An applied temperature is prescribed at the nodes, but thermal strain is computed based on element temperature. Element temperatures are obtained by averaging the temperatures of the nodes connected to a given element.

It is important to note that there is zero thermal strain applied at the reference temperature. The thermal strain reference temperature is stored in the element field named ‘thermal_strain_reference_temperature’ and may vary from element to element. Unless otherwise specified, the thermal strain reference temperature for each element is given by the initial temperature for that element, i.e., the model without other loads is in a stress-free state at the initial temperature. If desired, a different stress-free reference temperature can be used by prescribing the reference temperature with the `INITIAL CONDITION` command block as described in Section 7.3.

A second critical note is that thermal strain is applied based the change in temperature over a step. First the temperature is computed at the beginning of the step and the end of the step. Next the thermal strain is computed at these beginning and end of the step temperatures. The thermal strain increment is computed by the difference of these beginning and end of step strains. Finally this strain increment is applied to the material. Due to this thermal strain enforcement process the thermal strain is affected only by the slope of the thermal strain function. A vertical offset of the thermal strain function has no effect. Additionally if two discontinuous temperature functions are applied in different time periods thermal strain will not see the temperature jump between these two functions.

5.1.4.3 Thermal Strain Example

Example Consider the following thermal strain function:

```
BEGIN FUNCTION thermStrain
  TYPE IS PIECEWISE LINEAR
  BEGIN VALUES
    500  -0.04
    1000  0.01
    1500  0.03
  END
END
```

Let an element have an initial temperature of 1000 and a thermal strain reference temperature of 1000. Initially the element has zero applied thermal strain as it is at the strain-free reference temperature. If the element heats to temperature 1500, the thermal strain applied to the element will be 0.02 (the thermal strain at 1500 minus the thermal strain at the reference temperature 1000). If the element cools to 500, the thermal strain applied to the element will be -0.05 (the thermal strain at 500 minus the thermal strain at the reference temperature 1000).

5.1.5 Elastic Constants

```
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
```

Many material models contain input for basic elastic constants. These elastic constants may define the material response before yield, the elastic unloading response, or the approximate elastic stiffness of the material model. There exist several inter-related ways to define the elastic material properties. Six separate elastic constants may potentially be set YOUNGS MODULUS (E), POISSONS RATIO (ν), BULK MODULUS (K), SHEAR MODULUS (G), LAMBDA (λ), and TWO MU (2μ).

Generally elastic constants may either be set within the `BEGIN MATERIAL` command block or the `BEGIN PARAMETERS FOR MODEL . . . .` command block. Constants defined in the `BEGIN MATERIAL` block apply to all models in the material block. Constants defined in the `BEGIN PARAMETERS FOR MODEL . . . .` block apply to just that specific material model.

The parsed elastic constants are echoed to the log file when reading the material to verify correct setup. For example:

```
MATERIAL ELASTIC CONSTANT:steel_ELASTIC

E           Nu           G           K           Lambda
7.43e+11    0.34         2.77e+11  7.74e+11  5.89e+11
```

Note, the elastic constants printed at this point in the log are based off of the raw values of the global elastic constants such as YOUNGS MODULUS. The printed constants are not scaled for temperature dependence or any other external scaling factors. The proper element-by-element scaling will ultimately be used by the element mechanics and can be viewed by plotting the element variables `dilmod` (Dilatational Modulus) and `shrmmod` (Shear Modulus.)

Examples:

```
#
# Define elastic constants for a specific material model
#
```



```

BEGIN MATERIAL steel
  BEGIN PARAMETERS FOR MODEL ELASTIC
    YOUNGS MODULUS = 200e+9
    POISSONS RATIO = 0.3
  END
END

#
# Define elastic constants for all associated material models
#
BEGIN MATERIAL aluminum
  YOUNGS MODULUS = 40e+9
  POISSONS RATIO = 0.35

  BEGIN PARAMETERS FOR MODEL ELASTIC
    . . . .
  END
  BEGIN PARAMETERS FOR MODEL ELASTIC_PLASTIC
    . . . .
  END
END

```

The elastic properties are defined as follows:

- **YOUNGS MODULUS** describes the tensile elasticity, how much an object strains when it is pulled with opposing forces. Young's modulus must be a non-negative number.
- **POISSONS RATIO** describes how much a material expands laterally in unconstrained compression. An incompressible material has a Poisson's ratio of 0.5. The valid range for Poisson's ratio is from -1 to 0.5.
- **BULK MODULUS** describes the volumetric elasticity, how much an object shrinks under isotropic compression. Bulk modulus must be a non-negative number.
- **SHEAR MODULUS** describes the resistance of a solid to shear forces. A freely flowing fluid would have zero shear modulus. Shear modulus must be a non-negative number.
- **LAMBDA** is the first Lamé elastic constant. Lambda can have any value though is generally positive for real materials.
- **TWO MU** is simply twice the shear modulus.

The elastic constants are interrelated and defining any two of these constants (with the exception of **TWO MU** and **SHEAR MODULUS**) fully defines the elastic properties of the material. Generally exactly two of these constants should be set for each material model. Note, **TWO MU** and **SHEAR MODULUS** define the same constant (**TWO MU** is just twice the **SHEAR MODULUS**). Thus **TWO MU** and **SHEAR MODULUS** should not be defined at the same time.

Given Young's Modulus E and Poisson's ratio (ν) the other four elastic constants would be set as follows:

$$\text{BULK MODULUS: } K = \frac{E}{3(1 - 2\nu)} \quad (5.2)$$

$$\text{SHEAR MODULUS: } G = \frac{E}{2(1 + \nu)} \quad (5.3)$$

$$\text{LAMBDA: } \lambda = \frac{E\nu}{(1 + \nu)(1 - 2\nu)} \quad (5.4)$$

$$\text{TWO MU: } 2\mu = \frac{E}{1 + \nu} \quad (5.5)$$

5.1.6 The Isotropic/Kinematic hardening BETA parameter

```
BETA = <real>betaVal(1.0)
```

Several plasticity model variants make use of the Isotropic/Kinematic hardening factor `BETA`. In a three-dimensional framework, hardening is the law that governs how the yield surface evolves in stress space. If the yield surface grows uniformly in stress space, the hardening is called isotropic hardening. When `BETA` is 1.0 the model uses only isotropic hardening. If the yield surface does not grow in stress space but rather moves, the hardening is called kinematic hardening. When `BETA` is 0.0 the model uses only kinematic hardening. Values of `BETA` between 0.0 and 1.0 define a mixture of isotropic and kinematic hardening. Note, the input `BETA` has other meanings in some models, so see the specific material model documentation for details.

The mechanics of isotropic hardening are demonstrated in Figure 5.1.

The mechanics of kinematic hardening are demonstrated in Figure 5.2. If using pure kinematic hardening, then irrespective of the post yield hardening behavior the yield surface will never grow larger than the initial yield stress. Thus when using pure kinematic hardening the particular shape of the post yield hardening curve generally has no effect.

The mechanics of a mixed 0.5 `BETA` parameter is demonstrated in Figure 5.3.

5.1.7 Coordinate Systems

```
COORDINATE SYSTEM           = <string> coordinate_system_name
DIRECTION FOR ROTATION      = <real> 1|2|3
ALPHA                       = <real>  $\alpha_1$  (degrees)
SECOND DIRECTION FOR ROTATION = <real> 1|2|3
SECOND ALPHA                 = <real>  $\alpha_2$  (degrees)
```

Several anisotropic materials can be tied to a an element specific coordinate frame. By default the material R, S, and T coordinate frame lines up with the global X, Y, and Z coordinate system. A different material R, S, and T coordinate frame can be defined with the above commands, for example to define material properties that vary in the hoop and radial directions of a cylinder.

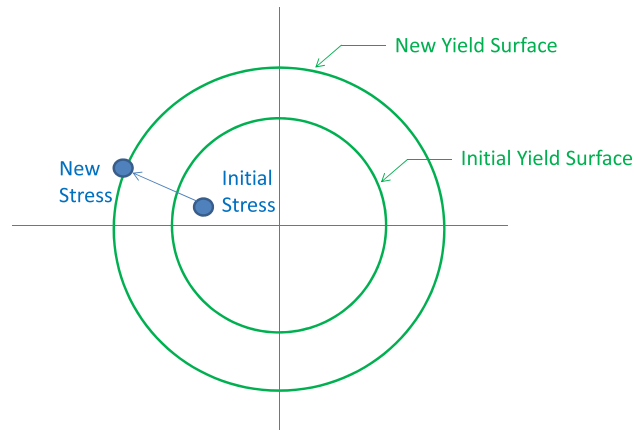


Figure 5.1: Isotropic Hardening, Beta = 1.0

Note, in addition to the coordinate system commands listed here, some material models have other model specific coordinate system options.

The specification of the principal material directions begins with the selection of a user-specified coordinate system given by the `COORDINATE SYSTEM` command line. The `coordinate_system_name` references a coordinate system defined by a separate `COORDINATE SYSTEM` command blocks (see Section 2.1.9). `COORDINATE SYSTEM` defines a local R, S, and T frame for each element based on the element centroid. This initial coordinate system can then be rotated twice to give the final material directions at each material point.

The first rotation of the initial coordinate system is defined using the `DIRECTION FOR ROTATION` and `ALPHA` command lines. The axis for rotation of the initial coordinate system is specified by the `DIRECTION FOR ROTATION` command line, 1 corresponds to the initial coordinate system local R axis, 2 corresponds to the initial coordinate system local S axis, and 3 corresponds to the initial coordinate system local T axis. The right hand rule angle of rotation about this axis is given by `ALPHA`. This rotation yields an intermediate coordinate system.

A secondary rotation of the intermediate coordinate system may be defined using the `SECOND DIRECTION FOR ROTATION` and `SECOND ALPHA` command lines. The axis for rotation of the intermediate coordinate system is specified by the `SECOND DIRECTION FOR ROTATION` command line, 1 corresponds to the intermediate coordinate system local R axis, 2 corresponds to the intermediate coordinate system local S axis, and 3 corresponds to the intermediate coordinate system local T axis. The right hand rule angle of rotation about this axis is given by `SECOND ALPHA`. The resulting coordinate system gives the final R, S, and T coordinate frame for the material directions.

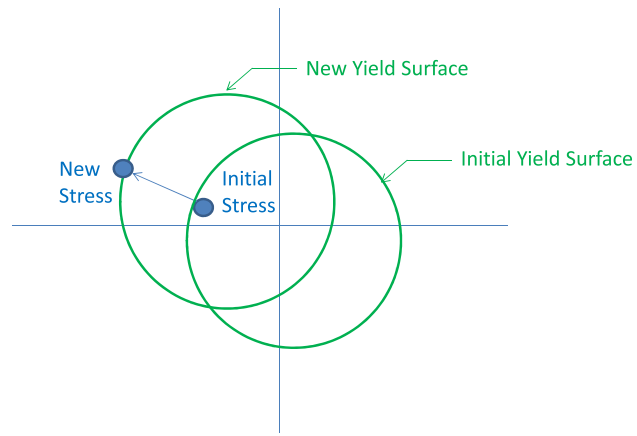


Figure 5.2: Kinematic Hardening, $\text{Beta} = 0.0$

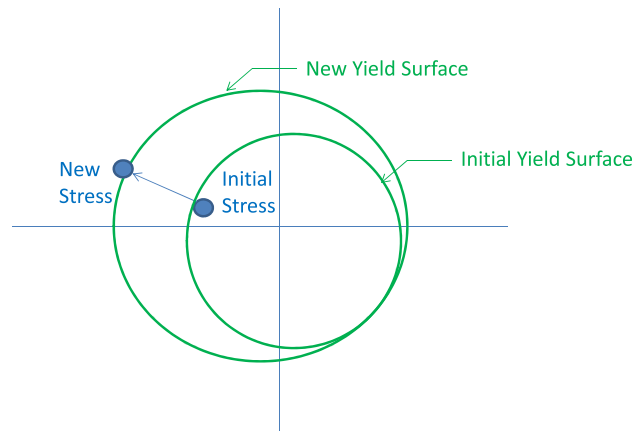


Figure 5.3: Mixed Hardening, $\text{Beta} = 0.5$

5.2 Material Models

This section contains descriptions of the general fully supported materials available in Sierra/SM. Additional models relevant to shock problems are described in the separate *Sierra/SMUser's Guide: Addendum for Shock Capabilities* document. Also additional experimental, in development, and specialty models may be found *Sierra/SMCapabilities in Development* manual.

5.2.1 Elastic Model

```
BEGIN PARAMETERS FOR MODEL ELASTIC
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU         = <real> 2μ
END [PARAMETERS FOR MODEL ELASTIC]
```

The elastic model is a hypoelastic extension of isotropic, small-strain, linear elasticity [2] [3] [4]. The stress-strain response for an isotropic, elastic material is

$$\sigma_{ij} = \lambda \delta_{ij} \varepsilon_{kk} + 2\mu \varepsilon_{ij} \quad (5.6)$$

where the Lamé constants, λ and μ , are given by

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)} \quad ; \quad \mu = \frac{E}{2(1+\nu)} \quad (5.7)$$

This model is extended to a finite-deformation, hypoelastic model by first making it a rate equation. Then the stress rate is replaced with an objective stress rate and the strain rate is replaced with the rate of deformation. This gives us

$$\overset{\circ}{\sigma}_{ij} = \lambda \delta_{ij} D_{kk} + 2\mu D_{ij} \quad (5.8)$$

The stress rate is arbitrary, as long as it is objective. Two objective stress rates are commonly used: the Jaumann rate and the Green-McInnis rate. For problems with fixed principal axes of deformation, these two rates give the same answers. For problems where the principal axes of deformation rotate during the deformation, the two rates can give different answers. Generally speaking there is no reason to pick one objective rate over another. Sierra/SM uses the Green-McInnis rate.

The fourth-order elastic moduli are used in many constitutive models. There are many equivalent representations for the elastic moduli. In index notation we present the following three representations

$$\overset{\circ}{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}$$

$$\mathbb{C}_{ijkl} = \frac{E}{1+\nu} \left[\frac{\nu}{1-2\nu} \delta_{ij} \delta_{kl} + \frac{1}{2} (\delta_{ik} \delta_{jl} + \delta_{il} \delta_{jk}) \right] \quad (5.9)$$

$$\mathbb{C}_{ijkl} = \lambda \delta_{ij} \delta_{kl} + \mu (\delta_{ik} \delta_{jl} + \delta_{il} \delta_{jk}) \quad (5.10)$$

$$\mathbb{C}_{ijkl} = K \delta_{ij} \delta_{kl} + \mu \left(\delta_{ik} \delta_{jl} + \delta_{il} \delta_{jk} - \frac{2}{3} \delta_{ij} \delta_{kl} \right) \quad (5.11)$$

where K is the elastic bulk modulus and is given by

$$K = \frac{E}{3(1-2\nu)} \quad (5.12)$$

See Section 5.1.5 for more information on elastic constants input.

There are no output variables available for the elastic model. For information about the elastic model, consult [5].

5.2.2 Thermoelastic Model

```
BEGIN PARAMETERS FOR MODEL THERMOELASTIC
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU        = <real>  $2\mu$ 
#
# Thermoelastic functions
#
YOUNGS MODULUS FUNCTION = <string> ym_function_name
POISSONS RATIO FUNCTION = <string> pr_function_name
END [PARAMETERS FOR MODEL THERMOELASTIC]
```

The thermoelastic material model is used to describe a temperature-dependent linear elastic material. This is a hypoelastic model generally valid for small strains.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The `YOUNGS MODULUS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope. The function defines a scale factor on Young's modulus as a function of temperature.
- The `POISSONS RATIO FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope. The function defines a scale factor on Poisson's ratio as a function of temperature.

For information about the thermoelastic model, consult [5]. Several plasticity models share the same underlying implementation. Output state variables available for this model are listed in Table 5.9. Note, depending on options used (temperature dependence of properties, failure, etc.) some of these state variables may not be computed.

5.2.3 Neo-Hookean Model

```

BEGIN PARAMETERS FOR MODEL NEO_HOOKEAN
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU         = <real> 2μ
END [PARAMETERS FOR MODEL NEO_HOOKEAN]

```

The neo-Hookean model is a hyperelastic generalization of isotropic, small-strain linear elasticity. The stress-strain response for the neo-Hookean model may be determined from a free energy function - in this case the strain energy density, W . The form of the strain energy density ([6]) is

$$W(C_{ij}) = \frac{1}{2}K \left[\frac{1}{2} (J^2 - 1) - \ln J \right] + \frac{1}{2}\mu (\bar{C}_{kk} - 3) \quad , \quad (5.13)$$

where K and μ are the bulk and shear moduli, respectively. The deformation measure is given by C_{ij} , the components of the right Cauchy-Green tensor, where $C_{ij} = F_{ki}F_{kj}$. The determinant of the deformation gradient is given by J and is a measure of the volumetric part of the deformation. $\bar{C}_{ij}$ provides the isochoric part of the deformation and is given by

$$\bar{C}_{ij} = \bar{F}_{ki}\bar{F}_{kj}, \quad ; \quad \bar{F}_{ij} = J^{-1/3}F_{ij} \quad . \quad (5.14)$$

The second Piola-Kirchhoff stress, with components S_{ij} , may be determined by taking a derivative of the strain energy density and the Cauchy stress may be found by mapping from the second Piola-Kirchhoff stress. The components of the Cauchy stress are

$$\sigma_{ij} = \frac{1}{2}K \left(J - \frac{1}{J} \right) \delta_{ij} + J^{-5/3}\mu \left(B_{ij} - \frac{1}{3}B_{kk}\delta_{ij} \right) \quad , \quad (5.15)$$

where $B_{ij} = F_{ik}F_{jk}$, are the components of the left Cauchy-Green tensor and δ_{ij} is the Kronecker delta.

Linearizing (5.15) we recover small strain linear elasticity

$$\begin{aligned} \sigma_{ij} &= \left(K - \frac{2}{3}\mu \right) u_{k,k}\delta_{ij} + \mu (u_{i,j} + u_{j,i}) \\ &= \left(K - \frac{2}{3}\mu \right) \varepsilon_{kk}\delta_{ij} + 2\mu\varepsilon_{ij} \quad . \end{aligned} \quad (5.16)$$

The neo-Hookean model is used for the recoverable (elastic) part for a number of inelastic, finite deformation constitutive models.

There are no output variables available for the neo-Hookean model.

5.2.4 Elastic Fracture Model

```
BEGIN PARAMETERS FOR MODEL ELASTIC_FRACTURE
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
# Fracture parameters
#
MAX STRESS                      = <real>  $\sigma_{max}$ 
CRITICAL CRACK OPENING STRAIN = <real>  $\epsilon_{ccos}$ 
END [PARAMETERS FOR MODEL ELASTIC_FRACTURE]
```

The elastic fracture model is linear elastic model with a simple failure model using a maximum-principal-stress failure criterion. Post failure, the stress decays isotropically based on additional strain parallel to the maximum principal stress. The critical crack opening strain, ϵ_{ccos} is the value of the component of strain over which the stress is decayed to zero. This strain parameter can be adjusted so that failure is mesh independent.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The maximum principal stress at which failure occurs is defined with the `MAX STRESS` command line.
- The component of strain over which the stress decays to zero is defined with the `CRITICAL CRACK OPENING STRAIN` command line. This component of strain is aligned with the maximum-principal-stress direction at failure.

Output variables available for this model are listed in Table 5.1.

Table 5.1: State Variables for ELASTIC FRACTURE Model (Section 5.2.4)

Name	Variable Description
CRACK_OPENING_STRAIN	Critical value of opening strain
FAILURE_DIRECTION	Crack opening direction - vector
FAILURE_DIRECTION_X	Crack opening direction - x component
FAILURE_DIRECTION_Y	Crack opening direction - y component
FAILURE_DIRECTION_Z	Crack opening direction - z component
PRINCIPAL_STRESS	Value of maximum principal stress
MAX_RADIUS	Von Mises stress at failure
MAX_PRESS	Stress tensor pressure at failure
MAX_STRESS	Maximum principal stress (UQ)
CRITICAL_CRACK_OPENING_STRAIN	Critical crack opening strain (UQ)

5.2.5 Elastic-Plastic Model

```

BEGIN PARAMETERS FOR MODEL ELASTIC_PLASTIC
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Hardening Behavior
#
YIELD STRESS    = <real>  $\sigma_y$ 
BETA            = <real>  $\beta$  (1.0)
HARDENING MODULUS = <real>  $H'$ 
END [PARAMETERS FOR MODEL ELASTIC_PLASTIC]

```

The elastic-plastic model is a hypoelastic, rate-independent linear hardening plasticity model. The rate form of the constitutive equation assumes an additive split of the rate of deformation into an elastic and plastic part

$$D_{ij} = D_{ij}^e + D_{ij}^p \quad (5.17)$$

The stress rate only depends on the elastic strain rate in the problem

$$\overset{\circ}{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e \quad (5.18)$$

where $\mathbb{C}_{ijkl}$ are the components of the fourth-order, isotropic elasticity tensor.

The key to the model is finding the plastic rate of deformation. For associated flow the plastic rate of deformation is in a direction normal to the yield surface. The yield surface is given by

$$f(\sigma_{ij}, \alpha_{ij}, \bar{\epsilon}^p) = \phi(\sigma_{ij}, \alpha_{ij}) - \bar{\sigma}(\bar{\epsilon}^p) = 0 \quad (5.19)$$

where ϕ is the effective stress, α_{ij} are the components of the back stress (used with kinematic hardening), and $\bar{\sigma}$ is the hardening function which is a function of an internal state variable, the equivalent plastic strain $\bar{\epsilon}^p$. An example of such a yield surface (plotted in the deviatoric π -plane) is presented below in Figure 5.4. The isotropy of the yield surface is clearly evident.

For the elastic plastic model a linear hardening law is assumed

$$\bar{\sigma} = \sigma_y + H' \bar{\epsilon}^p \quad (5.20)$$

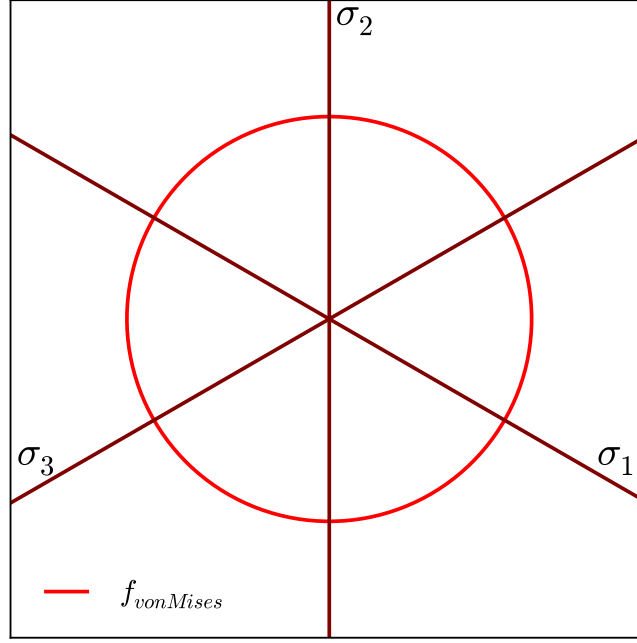


Figure 5.4: Example von Mises yield surface (J_2) used by the elastic-plastic model presented in the deviatoric π -plane. In this case the surface is plotted for $\alpha_{ij} = 0$ and $\bar{\epsilon}^p = 0$.

where σ_y is the yield stress and H' is the hardening modulus.

If the stress state is such that $f < 0$, the behavior of the material is elastic; if the stress state is such that $f = 0$ and $\dot{f} < 0$, i.e. the strain rate brings the stress inside the yield surface, then the behavior of the material is elastic; if the stress state is such that $f = 0$ and $\dot{f} > 0$, i.e. the strain rate brings the stress outside the yield surface, then plastic deformation occurs.

We assume associated flow in this model, which gives the plastic rate of deformation

$$D_{ij}^p = \dot{\gamma} \frac{\partial \phi}{\partial \sigma_{ij}} \quad (5.21)$$

where $\dot{\gamma}$ is the consistency parameter. For the elastic-plastic model the yield surface is assumed to be a von Mises yield surface with a back stress tensor to denote the center of the yield surface. The effective stress for a von Mises yield surface is

$$\phi(\sigma_{ij}) = \sqrt{\frac{3}{2} \xi_{ij} \xi_{ij}} \quad ; \quad \xi_{ij} = s_{ij} - \alpha_{ij} \quad (5.22)$$

where s_{ij} are the components of the deviatoric stress tensor

$$s_{ij} = \sigma_{ij} - \frac{1}{3} \delta_{ij} \sigma_{kk} \quad (5.23)$$

and α_{ij} are the components of the back stress tensor, another internal state variable.

The equivalent plastic strain is found through equating the rate of plastic work

$$\dot{W}^p = \sigma_{ij} D_{ij}^p = \bar{\sigma} \dot{\bar{\epsilon}}^p \rightarrow \dot{\bar{\epsilon}}^p = \dot{\gamma} \quad (5.24)$$

$$\bar{\epsilon}^p = \int_0^t \dot{\gamma} dt$$

Finally, the model allows for kinematic hardening through the back stress. The back stress is a symmetric, deviatoric rank two tensor that evolves in the following manner

$$\dot{\alpha}_{ij} = \frac{2}{3} (1 - \beta) H' D_{ij}^p \quad (5.25)$$

The radius of the yield surface can be defined, $R = \sqrt{\xi_{ij} \xi_{ij}}$. The evolution of the radius of the yield surface is given by

$$\dot{R} = \sqrt{\frac{2}{3}} \beta H' \dot{\bar{\epsilon}}^p \quad (5.26)$$

In (5.25) and (5.26) the parameter $\beta \in [0, 1]$ distributes the hardening between isotropic and kinematic hardening. If $\beta = 1$ the hardening is isotropic, if $\beta = 0$ the hardening is kinematic, and if β is between 0 and 1 the hardening is a combination of isotropic and kinematic.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input. The elastic constants describe both the pre-yield behavior of the model and the slope of post yield unloading.
- The yield stress, the stress at which yield first initiates, is defined with the `YIELD STRESS` command line.
- The hardening modulus, the slope of the post yield hardening curve, is defined with the `HARDENING MODULUS` command line.
- The beta parameter defines if hardening is isotropic or kinematic. See Section 5.1.6 for details on the beta parameter.

Output variables available for this model are listed in Table 5.2 and Table 5.3. For information about the elastic-plastic model, consult [5].

Table 5.2: State Variables for ELASTIC PLASTIC Model (Section 5.2.5)

Name	Description
EQPS	equivalent plastic strain, $\bar{\epsilon}^p$
RADIUS	radius of the yield surface, R
BACK_STRESS	back stress (symmetric tensor), α_{ij}

Table 5.3: State Variables for ELASTIC PLASTIC Model for Shells (Section 5.2.5)

Name	Description
EQPS	equivalent plastic strain, $\bar{\epsilon}^p$
TENSILE_EQPS	equivalent plastic strain only accumulated when the material is in tension (trace of stress tensor is positive)
RADIUS	radius of the yield surface, R
BACK_STRESS	back stress (symmetric tensor), α_{ij}
ITERATIONS	radial return iterations
ERROR	error in plane stress iterations
PS_ITER	plane stress iterations
TSTRAIN	integrated thickness strain

5.2.6 Elastic-Plastic Power-Law Hardening Model

```

BEGIN PARAMETERS FOR MODEL EP_POWER_HARD
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Hardening behavior
#
YIELD STRESS      = <real>  $\sigma_y$ 
HARDENING CONSTANT = <real>  $A$ 
HARDENING EXPONENT = <real>  $n$ 
LUDERS STRAIN     = <real>  $\varepsilon_L$ 
END [PARAMETERS FOR MODEL EP_POWER_HARD]

```

The elastic-plastic power law hardening model is a hypoelastic, rate-independent plasticity model with power law hardening [7]. The rate form of the constitutive equation assumes an additive split of the rate of deformation into an elastic and plastic part

$$D_{ij} = D_{ij}^e + D_{ij}^p \quad (5.27)$$

The stress rate only depends on the elastic strain rate in the problem

$$\dot{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e \quad (5.28)$$

where $\mathbb{C}_{ijkl}$ are the components of the fourth-order, isotropic elasticity tensor.

The key to integrating the model is finding the plastic rate of deformation. For associated flow the plastic rate of deformation is in a direction normal to the yield surface. The yield surface is given by

$$f(\sigma_{ij}, \bar{\varepsilon}^p) = \phi(\sigma_{ij}) - \bar{\sigma}(\bar{\varepsilon}^p) = 0 \quad (5.29)$$

where ϕ is the equivalent stress and $\bar{\sigma}$ is the hardening function which is a function of the equivalent plastic strain $\bar{\varepsilon}^p$. For this model the hardening function uses a power law

$$\bar{\sigma}(\bar{\varepsilon}^p) = \sigma_y + A \langle \bar{\varepsilon}^p - \varepsilon_L \rangle^n \quad (5.30)$$

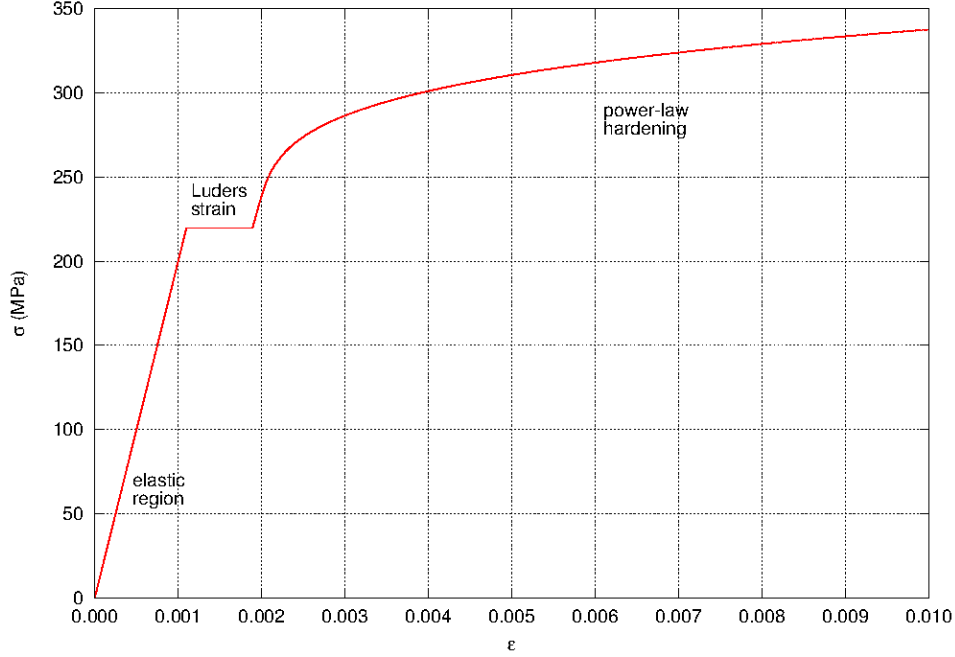


Figure 5.5: Typical stress-strain response for the power-law hardening model.

which is shown in Figure 5.5. The yield stress is σ_y , the hardening constant is A , the hardening exponent is n , and the Luder's strain is ε_L . The bracket $\langle \cdot \rangle$ is the Macaulay bracket defined as

$$\langle x \rangle = \begin{cases} 0, & \text{if } x \leq 0 \\ x, & \text{if } x > 0. \end{cases} \quad (5.31)$$

By assuming associated plastic flow, the plastic rate of deformation can be written as

$$D_{ij}^p = \dot{\gamma} \frac{\partial \phi}{\partial \sigma_{ij}}. \quad (5.32)$$

For this model the yield surface is chosen to be a von Mises yield surface, so

$$\phi(\sigma_{ij}) = \sqrt{\frac{3}{2} s_{ij} s_{ij}} \quad (5.33)$$

where s_{ij} are the components of the deviatoric stress

$$s_{ij} = \sigma_{ij} - \frac{1}{3} \delta_{ij} \sigma_{kk} \quad (5.34)$$

Unlike the elastic-plastic model 5.2.5, the power-law hardening model does not allow for kinematic hardening, so there is no back stress.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The `YIELD STRESS` is the stress at which the plastic power law yielding and hardening model takes effect. See Figure 5.5.
- The `LUDERS STRAIN` defines a regime of zero hardening modulus prior to onset of the power law hardening. A small Luder band is seen in the hardening behavior of many metals. See Figure 5.5 for details.
- The `HARDENING CONSTANT` command line and `HARDENING EXPONENT` command define the power law hardening curve. Past the Luder strain the hardened yield surface radius is given by the `HARDENING CONSTANT` times plastic strain to the `HARDENING EXPONENT` power.

Output variables available for this model are listed in Table 5.4 and Table 5.5. For information about the elastic-plastic power-law hardening model, consult [5].

Table 5.4: State Variables for EP POWER HARD Model (Section 5.2.6)

Name	Description
EQPS	equivalent plastic strain, $\bar{\epsilon}^p$
TENSILE_EQPS	equivalent plastic strain only accumulated when the material is in tension (trace of stress tensor is positive)
RADIUS	radius of yield surface, R
ITERATIONS	number of radial return iterations

Table 5.5: State Variables for EP POWER HARD Model for Shells (Section 5.2.6)

Name	Description
EQPS	equivalent plastic strain, $\bar{\epsilon}^p$
TENSILE_EQPS	equivalent plastic strain only accumulated when the material is in tension (trace of stress tensor is positive)
RADIUS	radius of yield surface, R
ITERATIONS	number of radial return iterations
ERROR	error in plane stress iterations
PS_ITER	plane stress iterations

5.2.7 Ductile Fracture Model

```

BEGIN PARAMETERS FOR MODEL DUCTILE_FRACTURE
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
# Yield surface parameters
#
YIELD STRESS      = <real>  $\sigma_y$ 
HARDENING CONSTANT = <real>  $A$ 
HARDENING EXPONENT = <real>  $n$ 
LUDERS STRAIN     = <real>  $\varepsilon_L$ 
#
# Failure parameters
#
CRITICAL TEARING PARAMETER = <real>  $t_p^{\text{crit}}$ 
CRITICAL CRACK OPENING STRAIN = <real>  $\varepsilon_{\text{ccos}}$ 
END [PARAMETERS FOR MODEL DUCTILE_FRACTURE]

```

The ductile fracture model is identical to the elastic-plastic power-law hardening model with the addition of a failure criterion and an isotropic decay of the stress to zero during the failure process within the constitutive model. To accomplish this task, the tearing parameter, t_p , proposed by Wellman [8] is introduced and the functional form as given as

$$t_p = \int_0^{\varepsilon} \left\langle \frac{2\sigma_{\max}}{3(\sigma_{\max} - \sigma_m)} \right\rangle^4 d\bar{\varepsilon}^p \quad (5.35)$$

where $\sigma_{\max}$ is the maximum principal stress, and σ_m is the mean stress. It can also be noted that the tearing parameter evolves during the plastic deformation regime as indicated by integrating over the effective plastic strain, $\bar{\varepsilon}^p$. The angle brackets denoting the Macaulay brackets, where

$$\langle x \rangle = \begin{cases} 0 & \text{if } x \leq 0 \\ x & \text{if } x > 0 \end{cases}, \quad (5.36)$$

are used to ensure that the failure process occurs only with tensile stress states and prevent “damage healing”. The failure process then initiates at a critical tearing parameter, t_p^{crit} , and the corresponding stress decay occurs over a strain interval corresponding to the *critical crack opening strain*, $\varepsilon_{\text{ccos}}$. Importantly, the $\varepsilon_{\text{ccos}}$ serves a dual role in that it may also be used to control the energy dissipated during failure. With respect to the latter point, careful selection of the critical crack opening

strain may be used to ensure consistent energy is dissipated through different meshes. This decay process is isotropic and linear with the current damage value being equivalent to the ratio of crack opening strain in the direction of the maximum principal stress to the critical value.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The `YIELD STRESS` is the stress at which the plastic power law yielding and hardening model takes effect. See Figure 5.5.
- The `LUDERS STRAIN` defines a regime of zero hardening modulus prior to onset of the power law hardening. A small Luder band is seen in the hardening behavior of many metals. See Figure 5.5 for details.
- The `HARDENING CONSTANT` command line and `HARDENING EXPONENT` command define the power law hardening curve. Past the Luder strain the hardened yield surface radius is given by the `HARDENING CONSTANT` times plastic strain to the `HARDENING EXPONENT` power.
- `CRITICAL TEARING PARAMETER` defines the t_p value at which fracture and subsequent decay of stress will occur.
- When the model undergoes additional strain after reaching the critical tearing parameter the stress in the model will decay to zero. The amount strain over which the stress decays to zero is defined with the `CRITICAL CRACK OPENING STRAIN` command line. The relevant opening strain is the component of strain that is aligned with the maximum-principal-stress direction at initial failure.

Output variables available for this model are listed in Table 5.6. For information about the ductile fracture material model, consult [8].

Table 5.6: State Variables for DUCTILE FRACTURE Model (Section 5.2.7)

Name	Description
EQPS	equivalent plastic strain, $\bar{\varepsilon}^p$
RADIUS	radius of yield surface, R
BACK_STRESS	back stress - tensor α_{ij}
TEARING_PARAMETER	Current value of the integrated tearing parameter
CRACK_OPENING_STRAIN	Current value of the crack opening strain. Will be zero prior to reaching the maximum tearing parameter.
FAILURE_DIRECTION	Crack opening direction (maximum principal stress direction at failure) - vector
DF_STRAIN_XX	XX component of current strain
DF_STRAIN_YY	YY component of current strain
DF_STRAIN_ZZ	ZZ component of current strain
DF_STRAIN_XY	XY component of current strain
DF_STRAIN_YZ	YZ component of current strain
DF_STRAIN_ZX	ZX component of current strain
MAX_RADIUS	Yield surface radius at failure
MAX_PRESS	Stress pressure norm at failure

5.2.8 Multilinear Elastic-Plastic Hardening Model

```

BEGIN PARAMETERS FOR MODEL MULTILINEAR_EP
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Hardening behavior
#
YIELD STRESS      = <real>  $\sigma_y$ 
BETA              = <real>  $\beta$  (1.0)
HARDENING FUNCTION = <string> hardening_function_name
#
# Functions
#
YOUNGS MODULUS FUNCTION = <string> ym_function_name
POISSONS RATIO FUNCTION = <string> pr_function_name
YIELD STRESS FUNCTION   = <string> yield_stress_function_name
END [PARAMETERS FOR MODEL MULTILINEAR_EP]

```

The multilinear elastic-plastic model is a generalization of the standard rate independent plasticity models already presented - the linear and power law hardening models. However, rather than having a specific functional form, the multilinear hardening model allows the user to input a piecewise linear function for the hardening curve. The rate form of the constitutive equation assumes an additive split of the rate of deformation into an elastic and plastic part such that

$$D_{ij} = D_{ij}^e + D_{ij}^p. \quad (5.37)$$

The stress rate only depends on the elastic strain rate so that,

$$\dot{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e, \quad (5.38)$$

where $\mathbb{C}_{ijkl}$ are the components of the fourth-order, isotropic elasticity tensor.

The key to the model is finding the plastic rate of deformation. For associated flow, the plastic rate of deformation is in the direction normal to the yield surface. With a yield surface given by

$$\phi(\sigma_{ij}) - \bar{\sigma}(\bar{\epsilon}^p) = 0 \quad (5.39)$$

then the plastic rate of deformation can be written as

$$D_{ij}^p = \dot{\gamma} \frac{\partial \phi}{\partial \sigma_{ij}}. \quad (5.40)$$

For this model the yield surface is taken to be a von Mises yield surface, such that

$$\phi(\sigma_{ij}) = \sqrt{\frac{3}{2} s_{ij} s_{ij}} \quad (5.41)$$

where s_{ij} are the components of the deviatoric stress

$$s_{ij} = \sigma_{ij} - \frac{1}{3} \delta_{ij} \sigma_{kk}. \quad (5.42)$$

For simplicity it is easier to write (5.40) in terms of the normal to the yield surface

$$D_{ij}^p = \dot{\gamma} N_{ij} \quad ; \quad N_{ij} = \frac{\partial \phi}{\partial \sigma_{ij}} / \left\| \frac{\partial \phi}{\partial \sigma_{ij}} \right\| \quad (5.43)$$

The model also incorporates temperature dependence in that the elastic properties and the yield stress can be functions of temperature. This is not as general as having the yield curves depend on temperature. For that behavior the thermoelastic-plastic model can be used.

An example stress vs. plastic strain hardening curve is shown in Figure 5.6. This curve was generated for a loading case of uniaxial strain. In this case, the effective stress is the same as the uniaxial. Therefore, for use with the multilinear elastic-plastic model this curve would simply have to be discretized and used as input.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The beta parameter defines if hardening is isotropic or kinematic. See Section 5.1.6 for details on the beta parameter.
- `YIELD STRESS` defines the stress where plastic yielding first occurs.
- The `HARDENING FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope. The function describes the hardening behavior of the material as stress versus equivalent plastic strain. The x values of the function should be values of equivalent plastic strain while the y values of the function can be either the increment of stress over the yield stress or the actual stress at the corresponding equivalent plastic strain. Note the hardening function can have its first point defined at (0,0), or at (0, YIELD_STRESS). Either function definition behaves the same as only the slope of the hardening function between two strains is used by the model.

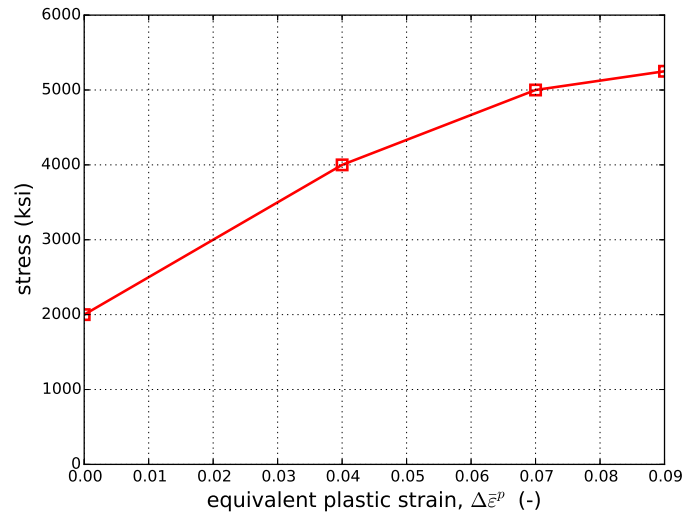


Figure 5.6: An example of a multilinear elastic-plastic stress-strain curve.

- The `YOUNGS MODULUS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Young's modulus as a function of temperature.
- The `POISSONS RATIO FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Poisson's ratio as a function of temperature.
- The `YIELD STRESS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on the yield stress as a function of temperature.

Output variables available for this model are listed in Table 5.7 and Table 5.8.

Table 5.7: State Variables for MULTILINEAR EP Model (Section 5.2.8)

Name	Description
EQPS	equivalent plastic strain
TENSILE_EQPS	equivalent plastic strain only accumulated when the material is in tension (trace of stress tensor is positive)
RADIUS	radius of yield surface
BACK_STRESS	back stress (symmetric tensor)
YOUNGS_MODULUS	the current Young's modulus as a function of temperature
POISSONS_RATIO	the current Poisson's ratio as a function of temperature
YIELD_STRESS	the current yield stress as a function of temperature
ITERATIONS	radial return iterations
YIELD_FLAG	inside (0) or on (1) the yield surface

Table 5.8: State Variables for MULTILINEAR EP Model for Shells (Section 5.2.8)

Name	Description
EQPS	equivalent plastic strain
TENSILE_EQPS	equivalent plastic strain only accumulated when the material is in tension (trace of stress tensor is positive)
RADIUS	radius of yield surface
BACK_STRESS	back stress (symmetric tensor)
YOUNGS_MODULUS	the current Young's modulus as a function of temperature
POISSONS_RATIO	the current Poisson's ratio as a function of temperature
YIELD_STRESS	the current yield stress as a function of temperature
ITERATIONS	radial return iterations
ERROR	error in plane stress iterations
PS_ITER	plane stress iterations

5.2.9 Multilinear Elastic-Plastic Hardening Model with Failure

```

BEGIN PARAMETERS FOR MODEL ML_EP_FAIL
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Hardening behavior
#
YIELD STRESS      = <real>  $\sigma_y$ 
BETA              = <real>  $\beta$  (1.0)
HARDENING FUNCTION = <string> hardening_function_name
#
# Functions
#
YOUNGS MODULUS FUNCTION = <string> ym_function_name
POISSONS RATIO FUNCTION = <string> pr_function_name
YIELD STRESS FUNCTION   = <string> yield_stress_function_name
#
# Failure parameters
#
CRITICAL TEARING PARAMETER      = <real>  $t_p^{\text{crit}}$ 
CRITICAL CRACK OPENING STRAIN = <real>  $\varepsilon_{\text{ccos}}$ 
CRITICAL BIAXIALITY RATIO      = <real> critical_ratio(0.0)
FAILURE EXPONENT                = <real>  $m$  (4.0)
END [PARAMETERS FOR MODEL ML_EP_FAIL]

```

Like the ductile fracture model, the multilinear elastic-plastic fail model is an extension of an existing plasticity model (multilinear elastic-plastic) to include a ductile failure criteria. Again, the tearing parameter criterion and failure propagation model of Wellman [8] is selected. Specifically, this approach uses a failure criterion (the tearing parameter, t_p) that is based on the history of the plastic strain and stress states. Most failure criteria for ductile failure involve some form of the stress triaxiality, or the ratio of the pressure and the effective (shear) stress. The tearing parameter, however, is slightly different in that it depends on the pressure and the maximum principal stress and is given as,

$$t_p = \int_0^\varepsilon \left\langle \frac{2\sigma_{\max}}{3(\sigma_{\max} - \sigma_m)} \right\rangle^m d\varepsilon_p, \quad (5.44)$$

with $\sigma_{\max}$ and σ_m being the maximum principal and mean stresses, respectively. The exponent m is typically taken to be 4 while the $\langle \cdot \rangle$ are Macaulay brackets defined as,

$$\langle x \rangle = \begin{cases} 0 & x \leq 0 \\ x & x > 0 \end{cases}, \quad (5.45)$$

and introduced so that failure only occurs and propagates under tensile stress states. Failure then initiates when the tearing parameter, t_p , reaches a critical value, t_p^{crit} . After this point, the stress decays (to 0) in a linear fashion according to the ratio of the crack opening strain in the maximum principal stress direction to its critical value, $\varepsilon_{\text{ccos}}$. Modification and control of this latter parameter is important as it may be used to ensure consistent energy is dissipated through different meshes.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The beta parameter defines if hardening is isotropic or kinematic. See Section 5.1.6 for details on the beta parameter.
- `YIELD STRESS` defines the stress for onset of yielding and plasticity.
- The `HARDENING FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope. The function describes the hardening behavior of the material as stress versus equivalent plastic strain. The x values of the function should be values of equivalent plastic strain while the y values of the function can be either the increment of stress over the yield stress or the actual stress at the corresponding equivalent plastic strain. Note the hardening function can have its first point defined at (0,0), or at (0, `YIELD_STRESS`). Either function definition behaves the same as only the slope of the hardening function between two strains is used by the model.
- The `YOUNGS MODULUS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Young's modulus as a function of temperature.
- The `POISSONS RATIO FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Poisson's ratio as a function of temperature.
- The `YIELD STRESS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on the yield stress as a function of temperature.
- `CRITICAL TEARING PARAMETER` defines the t_p value at which fracture and subsequent decay of stress will occur.
- When the model undergoes additional strain after reaching the critical tearing parameter the stress in the model will decay to zero. The amount strain over which the stress decays to zero is defined with the `CRITICAL CRACK OPENING STRAIN` command line. The relevant opening strain is the component of strain that is aligned with the maximum-principal-stress direction at initial failure.

- The `CRITICAL BIAXIALITY RATIO` command line should only be used under highly specific conditions and with extreme caution. It is intended only for the special case where the stress state is nearly biaxial, resulting in nearly identical principal strains. In this case, the eigenvector computation can give unreliable results for the direction vectors for the principal strains. If the ratio of the difference between two principal strains divided by their magnitude is less than the value specified by the `CRITICAL BIAXIALITY RATIO` command, the direction of the vector defining the crack opening strain will be given equal weight in each of the principal directions associated with those strains. The default value for the critical ratio is 0.0, which means that the principal directions will be accepted directly from the eigenvector computation. This command should only be used as a last resort if the loading is nearly biaxial and the default value has been demonstrated to lead to elements with high strains that are not failing long after reaching the critical tearing parameter.
- The `FAILURE EXPONENT` command line specifies the exponent on the tearing parameter, the m parameter in Equation 5.44. This exponent defaults to 4.0.

Output variables available for this model are listed in Table 5.9 and Table 5.10.

Table 5.9: State Variables for ML EP FAIL Model (Section 5.2.9)

Name	Variable Description
EQPS	Equivalent plastic strain
RADIUS	Radius of yield surface
BACK_STRESS	back stress - tensor
BACK_STRESS_XX	back stress - xx component
BACK_STRESS_YY	back stress - yy component
BACK_STRESS_ZZ	back stress - zz component
BACK_STRESS_XY	back stress - xy component
BACK_STRESS_YZ	back stress - yz component
BACK_STRESS_ZX	back stress - zx component
YOUNGS_MODULUS	Current Young's modulus as a function of temperature
POISSONS_RATIO	Current Poisson's ratio as a function of temperature
YIELD_STRESS	Current Yield stress as a function of temperature
TENSILE_EQPS	equivalent plastic strain only accumulated when the material is in tension (trace of stress tensor is positive)
ITERATIONS	radial return iterations
YIELD_FLAG	inside(0) or on(1) yield surface
TEARING_PARAMETER	Current integrated value of the tearing parameter. Zero until yield is reached
CRACK_OPENING_STRAIN	Current value of the crack opening strain. Zero until the critical tearing parameter is reached
FAILURE_DIRECTION	crack opening direction at failure - vector
FAILURE_DIRECTION_X	crack opening direction at failure - x component
FAILURE_DIRECTION_Y	crack opening direction at failure - y component
FAILURE_DIRECTION_Z	crack opening direction at failure - z component
MAX_RADIUS	maximum radius at initial failure
MAX_PRESSURE	maximum stress pressure norm at initial failure
CRITICAL_CRACK_OPENING_STRAIN	
CRITICAL_TEARING_PARAMETER	

Table 5.10: State Variables for ML EP FAIL Model for Shells(Section 5.2.9)

Name	Variable Description
EQPS	equivalent plastic strain
RADIUS	radius of yield surface
BACK_STRESS	back stress - tensor
BACK_STRESS_XX	back stress - xx component
BACK_STRESS_YY	back stress - yy component
BACK_STRESS_ZZ	back stress - zz component
BACK_STRESS_XY	back stress - xy component
BACK_STRESS_YZ	back stress - yz component
BACK_STRESS_ZX	back stress - zx component
YOUNGS_MODULUS	Current Young's modulus as a function of temperature
POISSONS_RATIO	Current Poisson's ratio as a function of temperature
YIELD_STRESS	Current Yield stress as a function of temperature
ITER	radial return iterations
ERROR	Error in plane stress iterations
PS_ITER	Plane stress iterations
TEARING_PARAMETER	Current integrated value of the tearing parameter. Zero until yield is reached
CRACK_OPENING_STRAIN	Current value of the crack opening strain. Zero until the critical tearing parameter is reached
FAILURE_DIRECTION	crack opening direction at failure - vector
FAILURE_DIRECTION_X	crack opening direction at failure - x component
FAILURE_DIRECTION_Y	crack opening direction at failure - y component
FAILURE_DIRECTION_Z	crack opening direction at failure - z component
RADIUS_MAX	maximum radius at initial failure
TENSILE_EQPS	equivalent plastic strain only accumulated when the material is in tension (trace of stress tensor is positive)

5.2.10 Elastic-Plastic Hardening Model with Failure

```
BEGIN PARAMETERS FOR MODEL ELASTIC_PLASTIC_FAIL
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
# Isotropic/Kinematic hardening factor
#
BETA           = <real>  $\beta$  (1.0)
YIELD STRESS   = <real>  $\sigma_y$ 
HARDENING FUNCTION = <string>hardening_function_name
POISSONS RATIO FUNCTION = <string>pr_function_name
YIELD STRESS FUNCTION = <string>yield_stress_function_name
YOUNGS MODULUS FUNCTION = <string>ym_function_name
#
# Failure parameters
#
CRITICAL TEARING PARAMETER = <real>crit_tearing
CRITICAL CRACK OPENING STRAIN = <real>critical_strain
END [PARAMETERS FOR MODEL ELASTIC_PLASTIC_FAIL]
```

This model behaves identical to the `ML_EP_FAIL` (Section [5.2.9](#)) material mode and only exists for legacy syntax compatibility purposes.

5.2.11 Thermoelastic-Plastic Model

```
BEGIN PARAMETERS FOR MODEL THERMOELASTIC_PLASTIC
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU        = <real> 2μ
#
# Isotropic/Kinematic hardening factor
#
BETA = <real>beta_parameter(1.0)
YIELD STRESS = <real>yield_stress
YOUNGS MODULUS FUNCTION = <string>ym_function_name
POISSONS RATIO FUNCTION = <string>pr_function_name
YIELD STRESS FUNCTION = <string>ys_function_name
TEMPERATURES = <real(s)>temperature_values
HARDENING FUNCTIONS = <string(s)>hardening_functions
END [PARAMETERS FOR MODEL THERMOELASTIC_PLASTIC]
```

The thermoelastic-plastic model is similar to the elastic-plastic model, but allows for temperature-dependent changes of the material properties. All elastic-plastic input parameters are still valid for the thermoelastic-plastic model, in addition to the temperature dependent functions.

In the above command blocks:

- See Section [5.1.5](#) for more information on elastic constants input.
- The beta parameter defines if hardening is isotropic or kinematic. See Section [5.1.6](#) for details on the beta parameter.
- The yield stress, the stress at which yield first initiates, is defined with the `YIELD STRESS` command line.
- The `YIELD STRESS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Yield Stress as a function of temperature.
- The `YOUNGS MODULUS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Young's modulus as a function of temperature.
- The `POISSONS RATIO FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Poisson's ratio as a function of temperature.
- The `TEMPERATURES` command line specifies the temperatures that correspond to the hardening functions defined in the `HARDENING FUNCTIONS` command line. The number of temperatures input here must match the number of hardening functions input.

- The `HARDENING FUNCTIONS` command line references the names of the functions defined in a `FUNCTION` command line in the SIERRA scope that describes the hardening behavior for the material as stress vs. equivalent plastic strain at the temperatures defined in the `TEMPERATURES` command line. The number of hardening functions input here must match the number of temperatures input. If the material temperature exactly matches one of the input temperatures the hardening of the material will exactly follow the hardening curve associated with that temperature. If the material temperature is bracketed by two hardening curves the actual hardening curve taken will be linearly interpolated between the bracketing curves. If the material temperature is either above the highest defined temperature function or below the lowest defined temperature function the the closest hardening curve will be used for the material hardening.

Several plasticity models share the same underlying implementation. Output state variables available for this model are listed in Table 5.9. Note, depending on options used (temperature dependence of properties, failure, etc.) some of these state variables may not be computed.

5.2.12 Thermoelastic-Plastic Model with Failure

```
BEGIN PARAMETERS FOR MODEL THERMOELASTIC_PLASTIC_FAIL
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU        = <real> 2μ
#
# Isotropic/Kinematic hardening factor
#
BETA = <real>beta_parameter(1.0)
TEMPERATURES = <real(s)>temperature_values
YIELD STRESS = <real>yield_stress
YOUNGS MODULUS FUNCTION = <string>ym_function_name
POISSONS RATIO FUNCTION = <string>pr_function_name
YIELD STRESS FUNCTION = <string>ys_function_name
HARDENING FUNCTIONS = <string(s)>hardening_functions
CRITICAL CRACK OPENING STRAIN = <real>critical_strain
CRITICAL TEARING PARAMETER = <real>crit_tearing
CRITICAL CRACK OPENING STRAIN FUNCTION = <string>crack_open_strain_function_name
CRITICAL TEARING PARAMETER FUNCTION = <string>tearing_parameter_function_name
END [PARAMETERS FOR MODEL THERMOELASTIC_PLASTIC_FAIL]
```

The thermoelastic-plastic model with failure is similar to the `ELASTIC_PLASTIC_FAIL` material model, but allows for additional temperature-dependent changes of the material.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The beta parameter defines if hardening is isotropic or kinematic. See Section 5.1.6 for details on the beta parameter.
- `YIELD STRESS` defines the stress for onset of yielding and plasticity.
- The `TEMPERATURES` command line specifies the temperatures that correspond to the hardening functions defined in the `HARDENING FUNCTIONS` command line. The number of temperatures input here must match the number of hardening functions input.
- The `HARDENING FUNCTIONS` command line references the names of the functions defined in a `FUNCTION` command line in the `SIERRA` scope that describes the hardening behavior for the material as stress vs. equivalent plastic strain at the temperatures defined in the `TEMPERATURES` command line. The number of hardening functions input here must match the number of temperatures input. If the material temperature exactly matches one of the input temperatures the hardening of the material will exactly follow the hardening curve associated with that temperature. If the material temperature is bracketed by two hardening curves the actual hardening curve taken will be linearly interpolated between the bracketing

curves. If the material temperature is either above the highest defined temperature function or below the lowest defined temperature function the the closest hardening curve will be used for the material hardening.

- The `POISSONS RATIO FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on Poisson's ratio as a function of temperature.
- The `YIELD STRESS FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on the yield stress as a function of temperature.
- `CRITICAL TEARING PARAMETER` defines the t_p value at which fracture and subsequent decay of stress will occur.
- When the model undergoes additionally strain after reaching the critical tearing parameter the stress in the model will decay to zero. The amount strain over which the stress decays to zero is defined with the `CRITICAL CRACK OPENING STRAIN` command line. The relevant opening strain is the component of strain that is aligned with the maximum-principal-stress direction at initial failure.
- The `CRITICAL CRACK OPENING STRAIN FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on the crack opening strain as a function of temperature.
- The `CRITICAL TEARING PARAMETER FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes a scale factor on the tearing parameter as a function of temperature.

Several plasticity models share the same underlying implementation. Output state variables available for this model are listed in Table 5.9. Note, depending on options used (temperature dependence of properties, failure, etc.) some of these state variables may not be computed.

5.2.13 Hosford Plasticity Model

```
BEGIN PARAMETERS FOR MODEL HOSFORD_PLASTICITY
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
# Yield surface parameters
#
YIELD STRESS = <real>  $\sigma_y$ 
A            = <real>  $a$  (1.0)
BETA        = <real>  $\beta$  (1.0)
#
# Hardening model
#
HARDENING MODEL = LINEAR | POWER_LAW | VOCE | USER_DEFINED |
    CUBIC_HERMITE_SPLIT | JOHNSON_COOK | POWER_LAW_BREAKDOWN
#
# Linear hardening
#
HARDENING MODULUS = <real>  $H'$ 
#
# Power-law hardening
#
HARDENING CONSTANT = <real>  $A$ 
HARDENING EXPONENT = <real>  $n$  (0.5)
LUDERS STRAIN      = <real>  $\epsilon_L$  (0.0)
#
# Voce hardening
#
HARDENING MODULUS      = <real>  $A$ 
EXPONENTIAL COEFFICIENT = <real>  $n$ 
#
# Johnson-Cook hardening
#
HARDENING FUNCTION = <string>hardening_function_name
RATE CONSTANT      = <real>  $C$ 
REFERENCE RATE      = <real>  $\dot{\epsilon}_0$ 
#
# Power law breakdown hardening
#
HARDENING FUNCTION = <string>hardening_function_name
```

```

RATE COEFFICIENT      = <real> g
RATE EXPONENT         = <real> m
#
# User defined hardening
#
HARDENING FUNCTION = <string>hardening_function_name
#
# Spline based hardening curve
#
CUBIC SPLINE TYPE    = <string>
CARDINAL PARAMETER   = <real> val
KNOT EQPS            = <real_list> vals
KNOT STRESS          = <real_list> vals
END [PARAMETERS FOR MODEL HOSFORD_PLASTICITY]

```

Like other elastic-plastic models in Lamé, the Hosford plasticity model is a rate-independent hypoelastic formulation. Unlike the Hill and other more complex plasticity models, it is isotropic. In a similar fashion to those models, the total rate of deformation is additively decomposed into an elastic and plastic part such that

$$D_{ij} = D_{ij}^e + D_{ij}^p. \quad (5.46)$$

The objective stress rate, depending only on the elastic deformation, may then be written as,

$$\overset{\circ}{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e. \quad (5.47)$$

The Hosford plasticity model utilizes a yield surface first put forth by W. F. Hosford in the 1970's [9] that is isotropic but non-quadratic. This specific form was proposed due to experimental observations of biaxial stretching in which neither the Tresca or J_2 yield surfaces could describe the results. In contrast to many of the yield surfaces proposed for similar purposes, only two parameters are utilized. Even with these limited terms, the developed model is quite versatile and can be reduced to von Mises or Tresca conditions as well as capturing responses in between. This yield surface is given as,

$$f(\sigma_{ij}, \bar{\epsilon}^p) = \phi(\sigma_{ij}) - \bar{\sigma}(\bar{\epsilon}^p) = 0, \quad (5.48)$$

in which $\phi(\sigma_{ij})$ is the Hosford effective stress and $\bar{\sigma}(\bar{\epsilon}^p)$ is the isotropic hardening function. The Hosford effective stress is a non-quadratic function of the principal stresses (σ_i , $i = 1, 2, 3$) and is given as

$$\phi(\sigma_{ij}) = \left[\frac{|\sigma_1 - \sigma_2|^a + |\sigma_2 - \sigma_3|^a + |\sigma_1 - \sigma_3|^a}{2} \right]^{1/a} \quad (5.49)$$

in which a is the yield surface exponent. Interestingly, if $a = 2$ or 4 the yield surface reduces to that of a J_2 von Mises surface while $a = 1$ or as $a \rightarrow \infty$ produces a Tresca like shape. If the value

of a is above 4 the yield surface takes a position between the Tresca and J_2 limits. Typical values are $a = 6$ or $a = 8$ for *bcc* and *fcc* metals, respectively [10]. To highlight this variability the yield surface

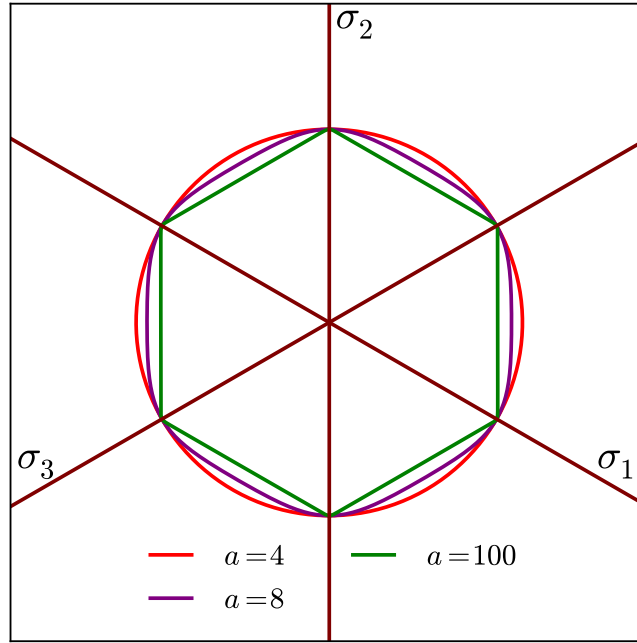


Figure 5.7: Example Hosford yield surfaces, $f(\sigma_{ij}, \bar{\epsilon}^p = 0; a)$, presented in the deviatoric π -plane. The presented surfaces correspond to the different yield exponents $a = 4, 8$, and 100 .

For the hardening function, $\bar{\sigma}(\bar{\epsilon}^p)$, a variety of forms including linear, power law, or a more general user defined function may be used.

An associated flow rule is utilized such that the plastic rate of deformation is normal to the yield surface and is given by,

$$\dot{D}_{ij}^p = \dot{\gamma} \frac{\partial \phi}{\partial \sigma_{ij}}, \quad (5.50)$$

where $\dot{\gamma}$ is the consistency multiplier enforcing $f = 0$ during plastic deformation. Given the form of f , it can also be shown that $\dot{\gamma} = \dot{\bar{\epsilon}}^p$.

For details, please see [11].

In the command blocks that define the Hosford plasticity model:

- See Section 5.1.5 for more information on elastic constants input.
- The reference nominal yield stress, $\bar{\sigma}$, is defined with the `YIELD STRESS` command line.
- The yield surface exponent, a , is defined with the `A` command line.

- The beta parameter defines if hardening is isotropic. See Section 5.1.6 for details on the beta parameter.
- The type of hardening law is defined with the `HARDENING MODEL` command line, other hardening commands then define the specific shape of that hardening curve.
- The hardening modulus for a linear hardening model is defined with the `HARDENING MODULUS` command line.
- The hardening constant for a power law hardening model is defined with the `HARDENING CONSTANT` command line.
- The hardening exponent for a power law hardening model is defined with the `HARDENING EXPONENT` command line.
- The Luder's strain for a power law hardening model is defined with the `LUDERS STRAIN` command line.
- The hardening function for a user defined hardening model is defined with the `HARDENING FUNCTION` command line.
- The shape of the spline for the spline based hardening is defined by the `CUBIC SPLINE TYPE`, `CARDINAL PARAMETER`, `KNOT EQPS`, and `KNOT STRESS` command lines.

Output variables available for this model are listed in Table 5.11.

Table 5.11: State Variables for HOSFORD PLASTICITY Model (Section 5.2.13)

Name	Description
EQPS	equivalent plastic strain, $\bar{\epsilon}^p$
EQDOT	equivalent plastic strain rate, $\dot{\bar{\epsilon}}^p$
SEFF	effective stress, ϕ

5.2.14 Hill Plasticity Model

```
BEGIN PARAMETERS FOR MODEL HILL_PLASTICITY
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
# Material coordinates system definition
#
COORDINATE SYSTEM           = <string> coordinate_system_name
DIRECTION FOR ROTATION      = <real> 1|2|3
ALPHA                       = <real>  $\alpha_1$  (degrees)
SECOND DIRECTION FOR ROTATION = <real> 1|2|3
SECOND ALPHA                = <real>  $\alpha_2$  (degrees)
#
# Yield surface parameters
#
YIELD STRESS = <real>  $\sigma_y$ 
R11 = <real>  $R_{11}$  (1.0)
R22 = <real>  $R_{22}$  (1.0)
R33 = <real>  $R_{33}$  (1.0)
R12 = <real>  $R_{12}$  (1.0)
R23 = <real>  $R_{23}$  (1.0)
R31 = <real>  $R_{31}$  (1.0)
#
# Hardening model
#
HARDENING MODEL = LINEAR | POWER_LAW | VOCE | USER_DEFINED |
  CUBIC_HERMITE_SPLIT | JOHNSON_COOK | POWER_LAW_BREAKDOWN
#
# Linear hardening
#
HARDENING MODULUS = <real>  $H'$ 
#
# Power-law hardening
#
HARDENING CONSTANT = <real>  $A$ 
HARDENING EXPONENT = <real>  $n$  (0.5)
LUDERS STRAIN      = <real>  $\epsilon_L$  (0.0)
#
# Voce hardening
#
```



```

HARDENING MODULUS          = <real> A
EXPONENTIAL COEFFICIENT    = <real> n
#
# Johnson-Cook hardening
#
HARDENING FUNCTION = <string>hardening_function_name
RATE CONSTANT           = <real> C
REFERENCE RATE           = <real>  $\dot{\epsilon}_0$ 
#
# Power law breakdown hardening
#
HARDENING FUNCTION = <string>hardening_function_name
RATE COEFFICIENT        = <real> g
RATE EXPONENT            = <real> m
#
# User defined hardening
#
HARDENING FUNCTION = <string>hardening_function_name
#
# Spline based hardening curve
#
CUBIC SPLINE TYPE        = <string>
CARDINAL PARAMETER       = <real> val
KNOT EQPS                 = <real_list> vals
KNOT STRESS               = <real_list> vals
END [PARAMETERS FOR MODEL HILL_PLASTICITY]

```

The Hill plasticity model is similar to other plasticity models except that it is not isotropic. It is a hypoelastic, rate-independent plasticity model. The rate form of the equation assumes an additive split of the rate of deformation into an elastic and plastic part

$$D_{ij} = D_{ij}^e + D_{ij}^p \quad (5.51)$$

The stress rate only depends on the elastic rate of deformation

$$\dot{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e \quad (5.52)$$

where $\mathbb{C}_{ijkl}$ are the components of the fourth-order, isotropic elasticity tensor.

The Hill plasticity model has an orthotropic yield surface that assumes orthogonal principal material directions. An example of this yield surface is presented below in Figure 5.8 along with examples of two isotropic surfaces – the von Mises (J_2) and Hosford (with $a = 8$). The various surface parameters correspond to 2090-T3 aluminum and the specific Hill strengths are found in [11]. By comparing the Hill surface to the two isotropic surfaces, the impact of the anisotropy

is clear. Additionally, substantial differences to the normals of the yield surfaces at points of intersection highlight the impact of the yield function selection on the resulting flow directions.

In functional form, the Hill yield surface looks like a von Mises yield surface except that there are 6 yield stresses: σ_{11}^y , σ_{22}^y , σ_{33}^y , τ_{12}^y , τ_{23}^y , and τ_{31}^y . These yield stresses correspond to 3 normal and 3 shear yield stresses. The yield function takes the form

$$\begin{aligned} \phi^2(\sigma_{ij}) = & F(\hat{\sigma}_{22} - \hat{\sigma}_{33})^2 + G(\hat{\sigma}_{33} - \hat{\sigma}_{11})^2 + H(\hat{\sigma}_{11} - \hat{\sigma}_{22})^2 \\ & + 2L\hat{\sigma}_{23}^2 + 2M\hat{\sigma}_{31}^2 + 2N\hat{\sigma}_{12}^2 = \bar{\sigma}^2(\bar{\epsilon}^p) \end{aligned} \quad (5.53)$$

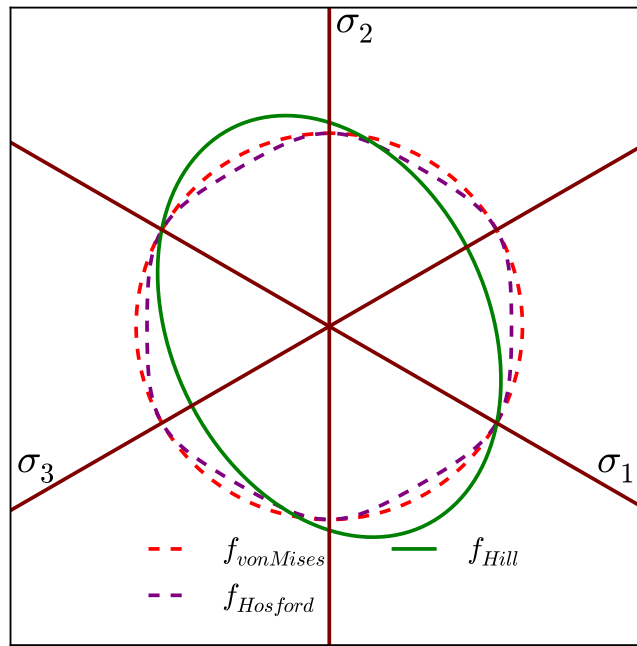


Figure 5.8: Example anisotropic Hill yield surface, $f_{Hill}(\sigma_{ij}, \bar{\epsilon}^p = 0)$, presented in the deviatoric π -plane fit to 2090-T3 aluminum. Comparison von Mises (J_2) and Hosford (with $a = 8$) surfaces are also presented.

The coefficients F , G , H , L , M , and N were introduced by Hill. In terms of the yield stresses they are:

$$\begin{aligned} F = \frac{(\bar{\sigma})^2}{2} \left[\frac{1}{(\sigma_{22}^y)^2} + \frac{1}{(\sigma_{33}^y)^2} - \frac{1}{(\sigma_{11}^y)^2} \right] ; \quad L = \frac{(\bar{\sigma})^2}{2} \left[\frac{1}{(\tau_{23}^y)^2} \right] \\ G = \frac{(\bar{\sigma})^2}{2} \left[\frac{1}{(\sigma_{33}^y)^2} + \frac{1}{(\sigma_{11}^y)^2} - \frac{1}{(\sigma_{22}^y)^2} \right] ; \quad M = \frac{(\bar{\sigma})^2}{2} \left[\frac{1}{(\tau_{31}^y)^2} \right] \end{aligned} \quad (5.54)$$

$$H = \frac{(\bar{\sigma})^2}{2} \left[\frac{1}{(\sigma_{11}^y)^2} + \frac{1}{(\sigma_{22}^y)^2} - \frac{1}{(\sigma_{33}^y)^2} \right] ; \quad N = \frac{(\bar{\sigma})^2}{2} \left[\frac{1}{(\tau_{12}^y)^2} \right]$$

where $\bar{\sigma}$ is a reference yield stress.

Rather than input the six independent yield stresses, the ratios of the yield stresses to some reference yield stress are generally used as input. These ratios are

$$\begin{aligned} R_{11} &= \frac{\sigma_{11}^y}{\bar{\sigma}} ; & R_{12} &= \sqrt{3} \frac{\tau_{12}^y}{\bar{\sigma}} \\ R_{22} &= \frac{\sigma_{22}^y}{\bar{\sigma}} ; & R_{23} &= \sqrt{3} \frac{\tau_{23}^y}{\bar{\sigma}} \\ R_{33} &= \frac{\sigma_{33}^y}{\bar{\sigma}} ; & R_{31} &= \sqrt{3} \frac{\tau_{31}^y}{\bar{\sigma}} \end{aligned} \tag{5.55}$$

These ratios are set up so that if $R_{ij} = 1$ then the yield surface is isotropic.

The orientation of the principal material axes with respect to the global Cartesian axes is defined in the same manner as the orthotropic elastic model. First a coordinate system is defined. For the Hill plasticity model it can be a rectangular or a cylindrical coordinate system. *The spherical coordinate system has not been implemented yet for this model.* After the coordinate system has been defined the material coordinate system can be defined through two successive rotations about axes in the rectangular or cylindrical coordinate system. In the case of the cylindrical coordinate system (and eventually the spherical coordinate system) this allows the principal material axes to vary point-wise in a given element block.

The plastic rate of deformation, as with the isotropic models, assumes associated flow

$$D_{ij}^p = \dot{\gamma} \frac{\partial \phi}{\partial \sigma_{ij}} \tag{5.56}$$

Given the form for ϕ , the consistency parameter, $\dot{\gamma}$ is equal to the rate of the equivalent plastic strain, $\dot{\bar{\epsilon}}^p$.

The hardening behavior is given by $\bar{\sigma}(\bar{\epsilon}^p)$. This hardening function can be a linear hardening function, a power law hardening function, or a user defined hardening function.

For more information about the Hill plasticity model, consult [12].

In the command blocks that define the Hill plasticity model:

- See Section 5.1.5 for more information on elastic constants input.
- See Section 5.1.7 for more information on material coordinates system definition commands.

- The reference nominal yield stress, $\bar{\sigma}$, is defined with the `YIELD STRESS` command line.
- The ratio of the normal yield stress in the $\bar{\mathbf{e}}_1\bar{\mathbf{e}}_1$ material direction is defined with the `R11` command line. The default is 1.0.
- The ratio of the normal yield stress in the $\bar{\mathbf{e}}_2\bar{\mathbf{e}}_2$ material direction is defined with the `R22` command line. The default is 1.0.
- The ratio of the normal yield stress in the $\bar{\mathbf{e}}_3\bar{\mathbf{e}}_3$ material direction is defined with the `R33` command line. The default is 1.0.
- The ratio of the shear yield stress in the $\bar{\mathbf{e}}_1\bar{\mathbf{e}}_2$ material direction is defined with the `R12` command line. The default is 1.0.
- The ratio of the shear yield stress in the $\bar{\mathbf{e}}_2\bar{\mathbf{e}}_3$ material direction is defined with the `R23` command line. The default is 1.0.
- The ratio of the shear yield stress in the $\bar{\mathbf{e}}_3\bar{\mathbf{e}}_1$ material direction is defined with the `R31` command line. The default is 1.0.
- The type of hardening law is defined with the `HARDENING MODEL` command line, other hardening commands then define the specific shape of that hardening curve.
- The hardening modulus for a linear hardening model is defined with the `HARDENING MODULUS` command line.
- The hardening constant for a power law hardening model is defined with the `HARDENING CONSTANT` command line.
- The hardening exponent for a power law hardening model is defined with the `HARDENING EXPONENT` command line.
- The Luder's strain for a power law hardening model is defined with the `LUDERS STRAIN` command line.
- The hardening function for a user defined hardening model is defined with the `HARDENING FUNCTION` command line.
- The shape of the spline for the spline based hardening is defined by the `CUBIC SPLINE TYPE`, `CARDINAL PARAMETER`, `KNOT EQPS`, and `KNOT STRESS` command lines.

Output variables available for this model are listed in Table 5.12.

Table 5.12: State Variables for HILL PLASTICITY Model (Section [5.2.14](#))

Name	Description
EQPS	equivalent plastic strain, $\bar{\epsilon}^p$
EQDOT	equivalent plastic strain rate, $\dot{\bar{\epsilon}}^p$
SEFF	effective stress, ϕ

5.2.15 Barlat Plasticity Model

```

BEGIN PARAMETERS FOR MODEL BARLAT_PLASTICITY
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
# Material coordinates system definition
#
COORDINATE SYSTEM           = <string> coordinate_system_name
DIRECTION FOR ROTATION      = <real> 1|2|3
ALPHA                       = <real>  $\alpha_1$  (degrees)
SECOND DIRECTION FOR ROTATION = <real> 1|2|3
SECOND ALPHA                = <real>  $\alpha_2$  (degrees)
#
# Yield surface parameters
#
YIELD STRESS = <real>  $\sigma_y$ 
A            = <real>  $a$  (4.0)
CP12         = <real>  $c'_{12}$  (1.0)
CP13         = <real>  $c'_{13}$  (1.0)
CP21         = <real>  $c'_{21}$  (1.0)
CP23         = <real>  $c'_{23}$  (1.0)
CP31         = <real>  $c'_{31}$  (1.0)
CP32         = <real>  $c'_{32}$  (1.0)
CP44         = <real>  $c'_{44}$  (1.0)
CP55         = <real>  $c'_{55}$  (1.0)
CP66         = <real>  $c'_{66}$  (1.0)
CPP12        = <real>  $c''_{12}$  (1.0)
CPP13        = <real>  $c''_{13}$  (1.0)
CPP21        = <real>  $c''_{21}$  (1.0)
CPP23        = <real>  $c''_{23}$  (1.0)
CPP31        = <real>  $c''_{21}$  (1.0)
CPP32        = <real>  $c''_{32}$  (1.0)
CPP44        = <real>  $c''_{44}$  (1.0)
CPP55        = <real>  $c''_{55}$  (1.0)
CPP66        = <real>  $c''_{66}$  (1.0)
#
# Hardening model
#
HARDENING MODEL = LINEAR | POWER_LAW | VOCE | USER_DEFINED |
CUBIC_HERMITE_SPLIT | JOHNSON_COOK | POWER_LAW_BREAKDOWN

```

```

#
# Linear hardening
#
HARDENING MODULUS = <real>  $H'$ 
#
# Power-law hardening
#
HARDENING CONSTANT = <real>  $A$ 
HARDENING EXPONENT = <real>  $n$  (0.5)
LUDERS STRAIN      = <real>  $\epsilon_L$  (0.0)
#
# Voce hardening
#
HARDENING MODULUS      = <real>  $A$ 
EXPONENTIAL COEFFICIENT = <real>  $n$ 
#
# Johnson-Cook hardening
#
HARDENING FUNCTION = <string>hardening_function_name
RATE CONSTANT      = <real>  $C$ 
REFERENCE RATE      = <real>  $\dot{\epsilon}_0$ 
#
# Power law breakdown hardening
#
HARDENING FUNCTION = <string>hardening_function_name
RATE COEFFICIENT    = <real>  $g$ 
RATE EXPONENT       = <real>  $m$ 
#
# User defined hardening
#
HARDENING FUNCTION = <string>hardening_function_name
#
# Spline based hardening curve
#
CUBIC SPLINE TYPE   = <string>
CARDINAL PARAMETER  = <real> val
KNOT EQPS           = <real_list> vals
KNOT STRESS         = <real_list> vals
END [PARAMETERS FOR MODEL BARLAT_PLASTICITY]

```

The Barlat plasticity model is a hypoelastic, rate-independent plasticity model. The underlying yield surface is both anisotropic and non-quadratic [13]. With respect to the former, linear transformations of the deviatoric stress are used to capture texture and anisotropy effects. The rate form of this model assumes an additive split of the rate of deformation into an elastic and plastic part

$$D_{ij} = D_{ij}^e + D_{ij}^p. \quad (5.57)$$

The stress rate only depends on the elastic rate of deformation

$$\dot{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e \quad (5.58)$$

where $\mathbb{C}_{ijkl}$ are the components of the fourth-order, isotropic elasticity tensor.

To describe anisotropy in the yield-behavior, two linear transformation tensors, C'_{ijkl} and C''_{ijkl} , are introduced such that,

$$s'_{ij} = C'_{ijkl} s_{kl} \quad ; \quad s''_{ij} = C''_{ijkl} s_{kl}, \quad (5.59)$$

with s_{ij} being the deviatoric stress tensor ($s_{ij} = \sigma_{ij} - 1/3 \sigma_{kk} \delta_{ij}$) and s'_{ij} and s''_{ij} being transformed stresses. Two transformations are used to capture both the anisotropy of the yield surface and flow rule. In Voigt notation the two transformation tensors are given as,

$$[C'] = \begin{bmatrix} 0 & -c'_{12} & -c'_{13} & 0 & 0 & 0 \\ -c'_{21} & 0 & -c'_{23} & 0 & 0 & 0 \\ -c'_{31} & -c'_{32} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & c'_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & c'_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & c'_{66} \end{bmatrix} \quad (5.60)$$

$$[C''] = \begin{bmatrix} 0 & -c''_{12} & -c''_{13} & 0 & 0 & 0 \\ -c''_{21} & 0 & -c''_{23} & 0 & 0 & 0 \\ -c''_{31} & -c''_{32} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & c''_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & c''_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & c''_{66} \end{bmatrix}. \quad (5.61)$$

Alternatively, the transformed stresses may be written in terms of the Cauchy stress tensor as,

$$s'_{ij} = L'_{ijkl} \sigma_{kl} \quad ; \quad s''_{ij} = L''_{ijkl} \sigma_{kl}, \quad (5.62)$$

where $L'_{ijkl} = C'_{ijmn} \Pi_{mnkl}$ and $L''_{ijkl} = C''_{ijmn} \Pi_{mnkl}$. In this case, Π_{ijkl} is the symmetric deviatoric projection tensor and takes the form of,

$$\Pi_{ijkl} = \frac{1}{2} (\delta_{ik} \delta_{jl} + \delta_{il} \delta_{jk}) - \frac{1}{3} \delta_{ij} \delta_{kl}. \quad (5.63)$$

In reduced form,

$$[L'] = \frac{1}{3} \begin{bmatrix} c'_{12} + c'_{13} & -2c'_{12} + c'_{13} & c'_{12} - 2c'_{13} & 0 & 0 & 0 \\ -2c'_{21} + c'_{23} & c'_{21} + c'_{23} & c'_{21} - 2c'_{23} & 0 & 0 & 0 \\ -2c'_{31} + c'_{32} & c'_{31} - 2c'_{32} & c'_{31} + c'_{32} & 0 & 0 & 0 \\ 0 & 0 & 0 & 3c'_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & 3c'_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & 3c'_{66} \end{bmatrix}, \quad (5.64)$$

and an analogous expression may be written for L''_{ijkl} .

The yield surface, f , is given as,

$$f(\sigma_{ij}, \bar{\epsilon}^p) = \phi(\sigma_{ij}) - \bar{\sigma}(\bar{\epsilon}^p) = 0, \quad (5.65)$$

in which $\phi(\sigma_{ij})$ is the effective stress and $\bar{\sigma}(\bar{\epsilon}^p)$ is the (isotropic) hardening function. The effective stress is written in terms of the principal transformed stresses (s'_i and s''_i , respectively) and the yield surface exponent, a , as,

$$\begin{aligned} \phi(\sigma_{ij}) = & \left\{ \frac{1}{4} \left[|s'_1 - s''_1|^a + |s'_1 - s''_2|^a + |s'_1 - s''_3|^a \right. \right. \\ & + |s'_2 - s''_1|^a + |s'_2 - s''_2|^a + |s'_2 - s''_3|^a \\ & \left. \left. + |s'_3 - s''_1|^a + |s'_3 - s''_2|^a + |s'_3 - s''_3|^a \right] \right\}^{1/a}. \end{aligned} \quad (5.66)$$

An example of such a yield surface is given in Figure 5.9 along with examples of previously presented (von Mises, Hosford, Hill) surfaces. The presented Barlat surface corresponds to that of 2090-T3 aluminum first characterized by Barlat *et al.* [13]. In Figure 5.9, both the anisotropy and non-quadratic nature of the yield surface is evident leading to differing strengths and flow directions at various stresses from any of the other models.

The orientation of the principal material axes with respect to the global Cartesian axes is defined in the same manner as the orthotropic elastic model. First, a reference coordinate system is defined. For the Barlat plasticity model it can be a rectangular or a cylindrical coordinate system. *The spherical coordinate system has not been implemented yet for this model.* After the coordinate system has been defined the material coordinate system can be defined through two successive rotations about axes in the specified rectangular or cylindrical coordinate system. In the case of the cylindrical coordinate system (and eventually the spherical coordinate system) this allows the principal material axes to vary point-wise in a given element block.

The plastic rate of deformation, as with the isotropic models, assumes associated flow

$$D_{ij}^p = \dot{\gamma} \frac{\partial \phi}{\partial \sigma_{ij}}, \quad (5.67)$$

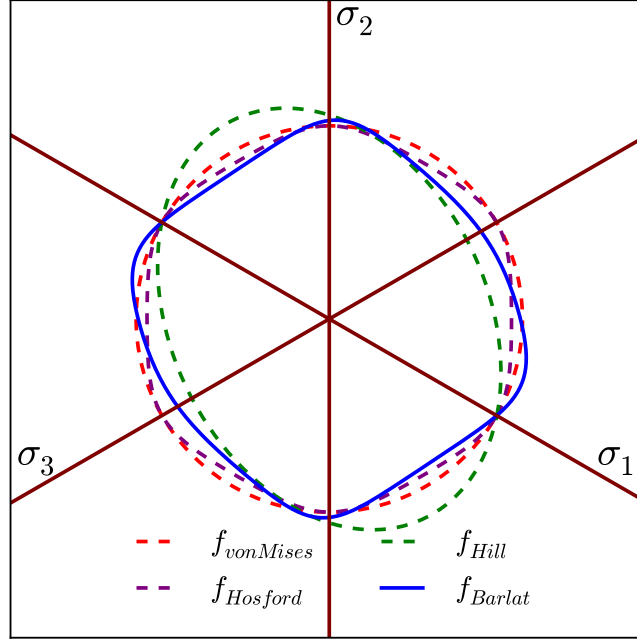


Figure 5.9: Example Barlat yield surface, $f_{Barlat}(\sigma_{ij}, \bar{\epsilon}^p = 0)$, of 2090-T3 aluminum presented in the deviatoric π -plane. Comparison von Mises (J_2), Hosford (with $a = 8$), and Hill surfaces are also presented for comparison.

in which $\dot{\gamma}$ is the consistency multiplier. Given the form for ϕ , $\dot{\gamma}$ is equal to the rate of the equivalent plastic strain, $\dot{\bar{\epsilon}}^p$. As the yield surface is cast in transformed stress space, determining the flow direction in Cartesian space may be done via the chain rule (details may be found in [11]) leading to an expression of the form,

$$\frac{\partial \phi}{\partial \sigma_{ij}} = \sum_{k=1}^3 \left(\frac{\partial \phi}{\partial s'_k} \frac{\partial s'_k}{\partial s'_{mn}} L'_{mnij} + \frac{\partial \phi}{\partial s''_k} \frac{\partial s''_k}{\partial s''_{mn}} L''_{mnij} \right). \quad (5.68)$$

The hardening behavior is given by $\bar{\sigma}(\bar{\epsilon}^p)$. This hardening function can be a linear hardening function, a power law hardening function, or a user defined hardening function.

For more information about the Barlat plasticity model, consult [13, 11].

In the command blocks that define the Barlat plasticity model:

- See Section 5.1.5 for more information on elastic constants input.
- See Section 5.1.7 for more information on material coordinates system definition commands.
- The reference nominal yield stress, $\bar{\sigma}$, is defined with the `YIELD STRESS` command line.
- The exponent for the yield surface description, a , is defined with the `A` command line.

- The transformation coefficient, c'_{12} , is defined with the CP12 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{13} , is defined with the CP13 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{21} , is defined with the CP21 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{23} , is defined with the CP23 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{31} , is defined with the CP31 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{32} , is defined with the CP32 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{44} , is defined with the CP44 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{55} , is defined with the CP55 command line. It is defaulted to 1.0.
- The transformation coefficient, c'_{66} , is defined with the CP66 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{12} , is defined with the CPP12 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{13} , is defined with the CPP13 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{21} , is defined with the CPP21 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{23} , is defined with the CPP23 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{31} , is defined with the CPP31 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{32} , is defined with the CPP32 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{44} , is defined with the CPP44 command line. It is defaulted to 1.0.
- The transformation coefficient, c''_{55} , is defined with the CPP55 command line. It is defaulted to 1.0.

- The transformation coefficient, c''_{66} , is defined with the `CPP66` command line. It is defaulted to 1.0.
- The type of hardening law is defined with the `HARDENING MODEL` command line, other hardening commands then define the specific shape of that hardening curve.
- The hardening modulus for a linear hardening model is defined with the `HARDENING MODULUS` command line.
- The hardening constant for a power law hardening model is defined with the `HARDENING CONSTANT` command line.
- The hardening exponent for a power law hardening model is defined with the `HARDENING EXPONENT` command line.
- The Luder's strain for a power law hardening model is defined with the `LUDERS STRAIN` command line.
- The hardening function for a user defined hardening model is defined with the `HARDENING FUNCTION` command line.
- The shape of the spline for the spline based hardening is defined by the `CUBIC SPLINE TYPE`, `CARDINAL PARAMETER`, `KNOT EQPS`, and `KNOT STRESS` command lines.

Output variables available for this model are listed in Table 5.13.

Table 5.13: State Variables for BARLAT PLASTICITY Model (Section 5.2.15)

Name	Description
EQPS	equivalent plastic strain, $\bar{\epsilon}^p$
EQDOT	equivalent plastic strain rate, $\dot{\bar{\epsilon}}^p$
SEFF	effective stress, ϕ

5.2.16 Johnson–Cook Model

```

BEGIN PARAMETERS FOR MODEL JOHNSON_COOK
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Yield surface parameters
#
YIELD STRESS = <real>  $\sigma_y$ 
HARDENING CONSTANT = <real>hardening_constant
HARDENING EXPONENT = <real>hardening_exponent
RHOCV = <real>rho_cv
RATE CONSTANT = <real>rate_constant
REFERENCE RATE = <real>reference_rate(0.001)
D1 = <real>d1(0.0)
D2 = <real>d2(0.0)
D3 = <real>d3(0.0)
D4 = <real>d4(0.0)
D5 = <real>d5(0.0)
EDOT_REF = <real>edot_ref(0.0)
#
# Temperature softening commands
#
BETA = <real>beta(0.95)
THERMAL EXPONENT = <real>thermal_exponent
REFERENCE TEMPERATURE = <real>reference_temperature
MELT TEMPERATURE = <real>melt_temperature
INITIAL TEMPERATURE = <real>init_temperature
FORMULATION = <int>form(0)
END [PARAMETERS FOR MODEL JOHNSON_COOK]

```

The Johnson–Cook model is used to model materials—typically metals—undergoing plastic deformation at high rates and finite strains. The hardening function is rate- and temperature-dependent and is defined by

$$\sigma = (\sigma_y + B\bar{\epsilon}_p^N)(1 + C \ln \dot{\epsilon}^*)(1 - T^{*M}), \quad (5.69)$$

where σ_y is the yield stress, B is the hardening constant, $\bar{\epsilon}_p$ is the equivalent plastic strain, N is the hardening exponent, and C is the rate constant. The non-dimensional effective total strain rate $\dot{\epsilon}^*$ is given by

$$\dot{\epsilon}^* = \dot{\epsilon}/\dot{\epsilon}_0, \quad (5.70)$$

where $\dot{\epsilon}_0$ is a reference strain rate defined by the input `EDOT_REF` material parameter. Note if `EDOT_REF` is undefined, then ϵ^* is set to the larger of either the maximum strain rate the material has encountered or the `REFERENCE RATE` input parameter. If `EDOT_REF` is defined, then the `REFERENCE RATE` input is not used by the model and has no effect.

Parameter T^* is the homologous temperature and is defined as

$$T^* = (T - T_{ref}) / (T_{melt} - T_{ref}), \quad (5.71)$$

where T is the current temperature, T_{ref} is the reference temperature, and T_{melt} is the melt temperature. In the case where $M \leq 0$, $(1 - (T^*)^M) = 1$.

Note that the temperature used internal to the Johnson–Cook model is *not* the standard model “temperature” field. Instead, the material temperature is defined via the `INITIAL TEMPERATURE` command. Plastic work internal to the model results in adiabatic heating and raises the material temperature. The resulting change in temperature is computed according to

$$\Delta T = \frac{\beta}{\rho C_v} \sigma_y \Delta \bar{\epsilon}_p, \quad (5.72)$$

where ρ is the material density, C_v is the specific heat, and β is the fraction of plastic work that is converted to heat, where $0 \leq \beta \leq 1$. By default, $\beta = 0.95$ though it may be set to a different value with the `BETA` command.

The Johnson–Cook model also encapsulates a failure criterion; the damage model [14] has a failure strain given by

$$\epsilon^f = (D_1 + D_2 \exp(D_3 \sigma^*)) (1 + D_4 \ln \dot{\epsilon}^*) (1 + D_5 T^*) \quad (5.73)$$

where D_1 , D_2 , D_3 , D_4 , and D_5 are material parameters, σ^* is the triaxiality, $\sigma^* = \sigma_m / \bar{\sigma}$, $\dot{\epsilon}^*$ is the dimensionless strain rate, and T^* is the homologous temperature, both defined above. Damage is accumulated over time:

$$D = \int_0^t \frac{\dot{\epsilon}^p}{\epsilon^f} dt. \quad (5.74)$$

When $D = 1$, the material has failed. By default, fracture behavior is not active. For more information about the Johnson–Cook material model, consult [15, 14].

5.2.16.1 Command lines

Valid command lines for a command block defining a Johnson–Cook model are as follows:

- Elastic constants: Consult Section 5.1.5 for information on elastic constants input.
- `YIELD STRESS` defines σ_y , the stress for onset of yield and plasticity.
- `HARDENING CONSTANT` defines B .
- `HARDENING EXPONENT` defines N .
- `RHOCV` defines ρC_v , the volumetric heat capacity.
- `INITIAL TEMPERATURE` defines the material initial temperature. Note that the Johnson–Cook material model temperature is *not* linked to the standard model “temperature” field that is set from `BEGIN PRESCRIBED TEMPERATURE` command blocks.
- `THERMAL EXPONENT` defines the thermal exponent M .

- REFERENCE TEMPERATURE defines the reference temperature T_{ref} .
- MELT TEMPERATURE defines the melt temperature T_{melt} .
- REFERENCE RATE defines the reference strain rate, $\dot{\epsilon}_0$. The default value is 0.001 s^{-1} .
- BETA defines the fraction of plastic work that is converted to heat, β . The default is 0.95.
- D1, D2, D3, D4, and D5 define fracture coefficients D_1 , D_2 , D_3 , D_4 , and D_5 , respectively. The default value for all coefficients is 0.0.
- FORMULATION controls the strain rate source term. A value of 0 triggers the use of the total strain rate, while 1 uses the plastic strain rate. The plastic strain rate may be stabler since it varies monotonically and at a much lower frequency than the total strain rate.

State variables for this model are listed in Table 5.14.

Table 5.14: State Variables for JOHNSON COOK Model (Section 5.2.16)

Name	Variable Description
RADIUS	radius of yield surface
EQPS	equivalent plastic strain
THETA	temperature
EQDOT	effective total strain rate
ITER	
EFAIL	failure strain, ϵ^f
DAMAGE	damage, D
YIELD_STRESS	yield stress

5.2.16.2 Warnings and Usage Guidelines



Warning: Strongly rate dependent models may fare poorly in implicit quasistatic solution. In implicit analyses the rate term used to evaluate the current load step is the rate seen by the model in the previous load step. This may cause the solution to oscillate between high- and low-rate equilibrium states from step to step.

5.2.17 BCJ Model

```
BEGIN PARAMETERS FOR MODEL BCJ
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
#
#
C1 = <real>c1
C2 = <real>c2
C3 = <real>c3
C4 = <real>c4
C5 = <real>c5
C6 = <real>c6
C7 = <real>c7
C8 = <real>c8
C9 = <real>c9
C10 = <real>c10
C11 = <real>c11
C12 = <real>c12
C13 = <real>c13
C14 = <real>c14
C15 = <real>c15
C16 = <real>c16
C17 = <real>c17
C18 = <real>c18
C19 = <real>c19
C20 = <real>c20
DAMAGE EXPONENT = <real>damage_exponent
INITIAL ALPHA_XX = <real>alpha_xx
INITIAL ALPHA_YY = <real>alpha_yy
INITIAL ALPHA_ZZ = <real>alpha_zz
INITIAL ALPHA_XY = <real>alpha_xy
INITIAL ALPHA_YZ = <real>alpha_yz
INITIAL ALPHA_XZ = <real>alpha_xz
INITIAL KAPPA = <real>initial_kappa
INITIAL DAMAGE = <real>initial_damage
YOUNGS MODULUS FUNCTION = <string>ym_function_name
POISSONS RATIO FUNCTION = <string>pr_function_name
SPECIFIC HEAT = <real>specific_heat
THETA OPT = <integer>theta_opt
```



```

    FACTOR = <real>factor
    RHO     = <real>rho
    TEMP0   = <real>temp0
END [PARAMETERS FOR MODEL BCJ]

```

The BCJ plasticity model is a state-variable model for describing the finite deformation behavior of metals. It uses a multiplicative decomposition of the deformation gradient into elastic, volumetric plastic, and deviatoric parts. The model considers the natural configuration defined by this decomposition and its associated thermodynamics. The model incorporates strain rate and temperature sensitivity, and damage, through a yield-surface approach in which state variables follow a hardening-minus-recovery format.

Because the BCJ model has such an extensive list of parameters, we will not present the usual synopsis of parameter names with command lines. As with most other material models, the `thermal strain` option is used to define thermal strains. See Section 5.1.4.1 and Section 5.1.4.2 for further information on defining and activating thermal strains. In addition, only two of the five elastic constants are required. The user should consult [16, 17, 18] for a description of the various parameters. The parameters for the `SPECIFIC HEAT`, `THETA OPT`, `FACTOR`, `RHO`, and `TEMP0` command lines are used to accommodate changes to the model for heat generation due to plastic dissipation. For coupled solid/thermal calculations, the plastic dissipation rate is stored as a state variable and passed to a thermal code as a heat source term. For uncoupled calculations, temperature is stored as a state variable, and temperature increases due to plastic dissipation are calculated within the material model.

If temperature is provided from an external source, `theta_opt` is set to 0. If the temperature is calculated by the BCJ model, `theta_opt` is set to 1.

Output variables available for this model are listed in Table 5.15.

5.2.17.1 Warnings and Usage Guidelines



Warning: Strongly rate-dependent models may fare poorly in implicit quasistatic solution. In implicit analyses the rate term used to evaluate the current load step is the rate seen by the model in the previous load step. This may cause the solution to oscillate between high- and low-rate equilibrium states from step to step.

Table 5.15: State Variables for BCJ Model

Name	Variable Description
BACK_STRESS_XX	back stress tensor - xx component
BACK_STRESS_YY	back stress tensor - yy component
BACK_STRESS_ZZ	back stress tensor - zz component
BACK_STRESS_XY	back stress tensor - xy component
BACK_STRESS_YZ	back stress tensor - yz component
BACK_STRESS_ZX	back stress tensor - zx component
KAPPA	hardening scalar
DAMAGE	damage term
DAMAGE_RATE	rate of change of damage term
EQPS	equivalent plastic strain
THETA	temperature for adiabatic heating
HEAT	rate of heating due to plastic dissipation
YM	Young's modulus
PR	Poisson's ratio

5.2.18 BCJ_MEM Model

```

BEGIN PARAMETERS FOR MODEL BCJ_MEM
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS = <real>  $G$ 
BULK MODULUS = <real>  $K$ 
LAMBDA = <real>  $\lambda$ 
TWO MU = <real>  $2\mu$ 

YOUNGS MODULUS FUNCTION = <string> youngs_modulus_scale_factor
POISSONS RATIO FUNCTION = <string> poissons_ratio_scale_factor
#
# Initial yield as function of temperature
#
RATE INDEPENDENT YIELD SHEAR COEFFICIENT = <real>  $Y_\mu$ 
RATE INDEPENDENT YIELD CONSTANT = <real>  $Y_0$ 
RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE = <real>  $Y_1$ 
RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE2 = <real>  $Y_2$ 
RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE3 = <real>  $Y_3$ 
RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE4 = <real>  $Y_4$ 
#
# Flow rule rate dependence
#
FLOW RULE COEFFICIENT CONSTANT = <real>  $f_1$ 
FLOW RULE COEFFICIENT TEMPERATURE DEPENDENCE = <real>  $f_2$ 
FLOW RULE EXPONENT CONSTANT = <real>  $n_1$ 
FLOW RULE EXPONENT TEMPERATURE DEPENDENCE = <real>  $n_2$ 
#
# Isotropic hardening
#
ISOTROPIC HARDENING SHEAR COEFFICIENT = <real>  $H_\mu$ 
ISOTROPIC HARDENING CONSTANT = <real>  $H_1$ 
ISOTROPIC HARDENING TEMPERATURE DEPENDENCE = <real>  $H_2$ 
ISOTROPIC DYNAMIC RECOVERY CONSTANT = <real>  $R_{d1}$ 
ISOTROPIC DYNAMIC RECOVERY TEMPERATURE DEPENDENCE = <real>  $R_{d2}$ 
ISOTROPIC STATIC RECOVERY CONSTANT = <real>  $R_{s1}$ 
ISOTROPIC STATIC RECOVERY TEMPERATURE DEPENDENCE = <real>  $R_{s2}$ 
ISOTROPIC STATIC RECOVERY SINH CONSTANT = <real>  $Q_1$ 
ISOTROPIC STATIC RECOVERY SINH TEMPERATURE DEPENDENCE = <real>  $Q_2$ 
#
# Kinematic hardening
#
BACK STRESS HARDENING CONSTANT = <real>  $h_1$ 
BACK STRESS HARDENING TEMPERATURE DEPENDENCE = <real>  $h_2$ 

```

```

BACK STRESS DYNAMIC RECOVERY CONSTANT = <real>  $r_{d1}$ 
BACK STRESS DYNAMIC RECOVERY TEMPERATURE DEPENDENCE =
    <real>  $r_{d2}$ 
#
# heat due to plastic dissipation
TEMPERATURE OPTION = <real>
PLASTIC DISSIPATION FACTOR = <real>  $\beta$ 
DENSITY FOR PLASTIC DISSIPATION CALCULATIONS = <real>  $\rho$ 
SPECIFIC HEAT FOR PLASTIC DISSIPATION CALCULATIONS =
    <real>  $c_p$ 
INITIAL TEMPERATURE FOR UNCOUPLED ADIABATIC HEATING =
    <real>  $\theta_o$ 
#
# damage evolution
DAMAGE EXPONENT = <real>  $m$ 
VOLUMETRIC EFFECTS OF DAMAGE OPTION = <real>
PRESSURE INTEGRATION OPTION = <real>
IMPLICIT DAMAGE SOLVER NUMBER OF ITERATIONS = <real>
IMPLICIT DAMAGE SOLVER RESIDUAL TOLERANCE = <real>
MAXIMUM CHANGE IN DAMAGE ALLOWED PER TIMESTEP = <real>
#
# misorientation variable
MISORIENTATION VARIABLE HARDENING CONSTANT = <real>  $h_\zeta$ 
MISORIENTATION VARIABLE HARDENING EXPONENT = <real>  $r$ 
#
# integration options
SEMI IMPLICIT PLASTIC STRAIN SOLVER NUMBER OF ITERATIONS =
    <real>
SEMI IMPLICIT PLASTIC STRAIN SOLVER RESIDUAL TOLERANCE =
    <real>
#
# initial values for state variables
#
INITIAL KAPPA = <real>  $\kappa_o$ 
INITIAL DAMAGE = <real>  $\phi_o$ 
INITIAL ZETA = <real>  $\zeta_o$ 
INITIAL ALPHA_XX = <real>  $\alpha_{xx_o}$ 
INITIAL ALPHA_XY = <real>  $\alpha_{xy_o}$ 
INITIAL ALPHA_XZ = <real>  $\alpha_{xz_o}$ 
INITIAL ALPHA_YY = <real>  $\alpha_{yy_o}$ 
INITIAL ALPHA_YZ = <real>  $\alpha_{yz_o}$ 
INITIAL ALPHA_ZZ = <real>  $\alpha_{zz_o}$ 
#
# Melting related variables
# Optional temperature at which material turns into a fluid and
# basic properties of that fluid
#

```

```

MELT TEMPERATURE = <real>melt_temp
FLUID VISCOSITY   = <real>fluid_viscosity
FLUID BULK        = <real>fluid_bulk
END [PARAMETERS FOR MODEL BCJ_MEM]

```

The BCJ_MEM material model, like BCJ, is a rate and temperature-dependent elasto-viscoplasticity model with isotropic damage. The two models have slightly different forms for the flow rule and static recovery expressions. In addition, the BCJ_MEM model includes the effects of recrystallization and grain growth [19]. Currently, the temperature must be prescribed in the input deck for these models to work. This will be changed in the future.

In its full form, the BCJ_MEM model has considerable complexity, but the material parameters in the list above are all optional, with the exception of two elastic constants and a yield strength. For example, in its simplest form, BCJ_MEM can be used to model elastic, perfectly-plastic materials by specifying the following:

```

BEGIN PARAMETERS FOR MODEL BCJ_MEM
  YOUNGS MODULUS = <real> E
  POISSONS RATIO = <real> ν
  RATE INDEPENDENT YIELD CONSTANT = <real> Y
END [PARAMETERS FOR MODEL BCJ_MEM]

```

To model an elastic-plastic material with linear hardening, the line

```

ISOTROPIC HARDENING CONSTANT = <real> H

```

would need to be added to specify the hardening rate. The more general form includes rate and temperature effects in the yield strength. BCJ_MEM is a state variable model that tracks history dependence through the use of internal state variables. Thus if the temperature and rate are not constant during loading, the yield stress is not a simple function of rate, temperature, and plastic strain. For the simplified case of uniaxial tension with no kinematic hardening, the following equations must be integrated to determine the current yield stress:

$$\dot{\sigma} = E(\dot{\epsilon} - \dot{\epsilon}_p), \quad (5.75)$$

$$\dot{\epsilon}_p = f(\theta) \sinh^{n(\theta)} \left\langle \frac{\sigma}{\kappa + Y(\theta)} - 1 \right\rangle, \quad (5.76)$$

$$\dot{\kappa} = \kappa \frac{\dot{\mu}}{\mu} + [H(\theta) - R_d(\theta)\kappa] \dot{\epsilon}_p - R_s(\theta)\kappa \sinh(Q(\theta)\kappa), \quad (5.77)$$

where θ is the temperature and κ represents the isotropic hardening variable, which evolves according to a hardening minus dynamic recovery form originally proposed by Kocks and Mecking [20], with a static recovery term developed by Nes [21]. The flow rule can be inverted to obtain the yield stress:

$$\sigma_y = [Y + \kappa] \left\{ 1 + \sinh^{-1} \left[\left(\frac{\dot{\epsilon}^p}{f} \right)^{1/n} \right] \right\}. \quad (5.78)$$

To illustrate the evolution of the yield stress for the simple case of constant strain rate, constant temperature loading with no stage IV hardening and no static recovery, the equation for the isotropic hardening variable reduces to

$$\dot{\kappa} = [H(\theta) - R_d(\theta)\kappa] \dot{\epsilon}_p \quad (5.79)$$

which can be integrated analytically; the flow rule can be inverted to get the following relation for yield strength:

$$\sigma_y = \left[Y + \frac{H}{R_d} (1 - e^{-R_d \epsilon_p}) \right] \left\{ 1 + \sinh^{-1} \left[\left(\frac{\dot{\epsilon}_p}{f} \right)^{1/n} \right] \right\} \quad (5.80)$$

The expression in brackets represents the sum of the rate-independent yield strength and the hardening due to plastic strain; the expression in braces provides the rate dependence of the yield strength. This equation can be compared to Equation (5.69) of the Johnson Cook model.

The temperature dependence of the initial yield strength is of the form

$$Y(\theta) = \frac{Y_o}{Y_4 + e^{-Y_1/\theta}} \frac{1}{2} [1 + \tanh(Y_2(Y_3 - \theta))], \quad (5.81)$$

where Y_i are material constants.

The yield can also be specified as a simple constant times the shear modulus,

$$Y(\theta) = Y_\mu \mu(\theta) \quad (5.82)$$

where the shear modulus can depend on temperature through user-defined scale factors:

```
YOUNGS MODULUS FUNCTION = <string> youngs_modulus_scale_factor
POISSONS RATIO FUNCTION = <string> poissons_ratio_scale_factor
```

Similarly, the temperature dependence of the hardening parameter in the evolution equation for the isotropic hardening variable can be specified in one of two ways:

$$H(\theta) = H_1 - H_2 \theta \quad (5.83)$$

or

$$H(\theta) = H_\mu \mu(\theta) \quad (5.84)$$

The other coefficients in the evolution equation for the isotropic hardening variable have an Arrhenius-type temperature dependence, as shown here for the dynamic recovery parameter:

$$R_d(\theta) = R_{d1} e^{-R_{d2}/\theta} \quad (5.85)$$

Rate effects in the flow rule: The flow rule coefficient has an Arrhenius temperature dependence:

$$f(\theta) = f_1 e^{-f_2/\theta}. \quad (5.86)$$

The exponent in the flow rule has the following temperature dependence:

$$n(\theta) = n_1 + \frac{n_2}{\theta}. \quad (5.87)$$

The rate dependence of the model can make the system of equations stiff. The default integration for plastic strain utilizes the radial return method. A semi-implicit integration scheme that is implicit in plastic strain can be selected by setting the `SEMI IMPLICIT PLASTIC STRAIN SOLVER NUMBER OF ITERATIONS` to whatever value is desired. If the residual does not drop below the tolerance specified by `SEMI IMPLICIT PLASTIC STRAIN SOLVER RESIDUAL TOLERANCE`, then the radial return value for plastic strain is used instead, and a message is sent to the output file to this effect. It is recommended to always use the semi-implicit plastic strain solver when running implicit analyses since it will improve convergence. For explicit, the errors incurred by not doing the semi-implicit integration are not significant enough to warrant the time cost of implicit iterations.

Stage IV hardening: Stage IV hardening is modeled through the misorientation variable, ζ , which loosely represents the geometrically necessary dislocation density and evolves according to the relation proposed by Kok et al. [22]. The misorientation variable, when used, adds an additional hardening term to the evolution equation for the isotropic hardening variable:

$$\dot{\kappa} = \kappa \frac{\dot{\mu}}{\mu} + \left[H(\theta) \left(1 + \frac{\zeta}{\kappa} \right) - R_d(\theta)\kappa \right] \dot{\epsilon}_p - R_s(\theta)\kappa \sinh(Q(\theta)\kappa) \quad (5.88)$$

The parameters in the evolution equation for the misorientation variable are independent of temperature, since they represent geometrically necessary dislocations:

$$\frac{d}{dt} \left(\frac{\zeta}{\mu} \right) = h_\zeta \left(\frac{\zeta}{\mu} \right)^{1-1/r} \dot{\epsilon}_p \quad (5.89)$$

Heat generation due to plastic dissipation: Heat generation due to plastic dissipation can be modeled in `BCJ_MEM` in two ways: for coupled solid/thermal calculations (`TEMPERATURE OPTION = 0`), the plastic dissipation rate is stored as a state variable (`AHEAT`) and passed to a thermal code as a heat source term; for uncoupled calculations (`TEMPERATURE OPTION = 1`), temperature is stored as a state variable (`THETA`), and temperature evolution due to adiabatic heating is calculated within the material model. The heat generated, `AHEAT`, is calculated as $\beta\sigma\dot{\epsilon}_p$. For uncoupled adiabatic heating, the temperature, `THETA`, evolves as

$$\dot{\theta} = \frac{\beta}{\rho c_p} \sigma \dot{\epsilon}_p \quad (5.90)$$

where β is the fraction of plastic work that is dissipated as heat, ρ is the density, and c_p is the specific heat.

Isotropic damage : Isotropic damage is included in the `BCJ_MEM` through a void growth equation proposed by Cocks and Ashby [23]:

$$\dot{\phi} = \sqrt{\frac{3}{2}} \dot{\epsilon}_p \frac{1 - (1 - \phi)^{m+1}}{(1 - \phi)^m} \sinh \left[\frac{2(2m - 1)}{2m + 1} \frac{p}{\sigma_{vm}} \right], \quad (5.91)$$

where σ_{vm} is the von Mises stress, p is the hydrostatic stress, and m is the `DAMAGE EXPONENT`. When damage is included in the model, the yield stress decreases with damage according to

$$\sigma_y = (1 - \phi)(Y + \kappa) \left\{ 1 + \sinh^{-1} \left[\left(\frac{\dot{\epsilon}_p}{f} \right)^{1/n} \right] \right\} \quad (5.92)$$

We assume Young's modulus and the shear modulus decrease as damage increases, according to $E(\theta, \phi) = E(\theta)(1 - \phi)$ and $\mu(\theta, \phi) = \mu(\theta)(1 - \phi)$. It then follows that the bulk modulus must vary in the same way:

$$K(\theta, \phi) = k(\theta)(1 - \phi) \quad (5.93)$$

Due to the effects of damage, the pressure evolution equation becomes:

$$\dot{p} = \left[\frac{\dot{k}(\theta)}{k(\theta)} - \frac{\dot{\phi}}{1 - \phi} \right] p + 3D_{ave}k(\theta)(1 - \phi) - k(\theta)\dot{\phi} \quad (5.94)$$

In this equation, the last term is due to volumetric expansion of the material as voids grow. This term can lead to some non-physical responses under certain loading conditions, so its inclusion is optional in the BCJ_MEM material model and is controlled by the `VOLUMETRIC EFFECTS OF DAMAGE OPTION` parameter. If this optional parameter is 0 (default), then the volumetric term is not included; if the value is 1, then the volumetric term is active. The integration of the pressure equation can be selected by setting the value of `PRESSURE INTEGRATION OPTION` to either 0 for implicit (Backward Euler) or 1 for implicit midpoint method.

The integration algorithm for damage solves the damage equation analytically using the midpoint value for triaxiality. The default value for `IMPLICIT DAMAGE SOLVER NUMBER OF ITERATIONS` is 50, but can be set to whatever value is desired. If the residual does not drop below the tolerance specified by `IMPLICIT DAMAGE SOLVER RESIDUAL TOLERANCE`, then the value for damage from the last iteration attempt is used instead, and a message is sent to the output file to this effect.

Modeling recrystallization: Recrystallization is a process by which the microstructure in a worked material is softened by the nucleation and growth of a new set of grains with relatively low dislocation density. The method used by BCJ_MEM to model recrystallization is fully documented in Reference [19], so it will not be repeated here. Additional output variables when modeling recrystallization are listed in Table 5.17

Kinematic hardening: The yield surface in stress space can either grow (or contract) through the isotropic hardening variable or translate through evolution of the back stress, or kinematic hardening, variable. The back stress evolves according to the equation

$$\dot{\alpha} = h(\theta)d^p - r_d(\theta)\dot{\epsilon}^p \sqrt{\frac{3}{2}}\|\alpha\|\alpha, \quad (5.95)$$

where

$$h(\theta) = h_1 - h_2\theta, \quad (5.96)$$

$$r_d(\theta) = r_{d1}e^{-r_{d2}/\theta}. \quad (5.97)$$

When kinematic hardening is desired, the hardening and recovery must be specified by the parameters h_1 , h_2 , r_{d1} , and r_{d2} .

State variable initialization: The initial values for the internal state variables are specified through the input values for κ_o , ϕ_o , ζ_o , and α_{ijo} . The default values for the initial values of κ_o , ζ_o and X_o are 1.0e-6 to avoid singularities in the evolution equations.

Table 5.16: State Variables for BCJ_MEM Model

Name	Variable Description
BACK_STRESS_XX	back stress tensor - xx component
BACK_STRESS_YY	back stress tensor - yy component
BACK_STRESS_ZZ	back stress tensor - zz component
BACK_STRESS_XY	back stress tensor - xy component
BACK_STRESS_YZ	back stress tensor - yz component
BACK_STRESS_ZX	back stress tensor - zx component
KAPPA	hardening scalar
DAMAGE	void volume fraction
DAMAGE_RATE	rate of change of void volume fraction
EQPS	equivalent plastic strain
THETA	temperature for adiabatic heating
AHEAT	plastic dissipation rate
EQPSDOT	plastic strain rate
SIGY_RT	room temperature yield stress at quasistatic conditions
IBOND	contact variable to switch from sliding to glued contact based on recrystallized volume fraction

Output variables available for this model are listed in Tables 5.16 and 5.17.

Further questions about the model, or requests for changes/enhancements, can be sent to aabrown@sandia.gov.

5.2.18.1 Warnings and Usage Guidelines



Warning: Strongly rate-dependent models may fare poorly in implicit quasistatic solution. In implicit analyses the rate term used to evaluate the current load step is the rate seen by the model in the previous load step. This may cause the solution to oscillate between high- and low-rate equilibrium states from step to step.

Table 5.17: Additional State Variables for BCJ_MEM Model When Modeling Recrystallization

Name	Variable Description
GRAIN_SIZE	grain size
REX_CYCLE_A	recrystallized volume fraction for cycle A
REX_CYCLE_B	recrystallized volume fraction for cycle B
REX_CYCLE_C	recrystallized volume fraction for cycle C
REX_CYCLE_D	recrystallized volume fraction for cycle D
BACK_STRESS_NO_REX_XX	xx component of back stress in un-recrystallized volume fraction
BACK_STRESS_NO_REX_YY	yy component of back stress in un-recrystallized volume fraction
BACK_STRESS_NO_REX_ZZ	zz component of back stress in un-recrystallized volume fraction
BACK_STRESS_NO_REX_XY	xy component of back stress in un-recrystallized volume fraction
BACK_STRESS_NO_REX_YZ	yz component of back stress in un-recrystallized volume fraction
BACK_STRESS_NO_REX_ZX	zx component of back stress in un-recrystallized volume fraction
SKAPP_NO_REX	isotropic hardening in the un-recrystallized volume fraction
SKAPP_CYCLE_A	isotropic hardening in the recrystallized volume fraction for cycle A
SKAPP_CYCLE_B	isotropic hardening in the recrystallized volume fraction for cycle B
SKAPP_CYCLE_C	isotropic hardening in the recrystallized volume fraction for cycle C
SKAPP_CYCLE_D	isotropic hardening in the recrystallized volume fraction for cycle D
ZETA_NO_REX	misorientation in the un-recrystallized volume fraction
ZETA_CYCLE_A	misorientation in the un-recrystallized volume fraction for cycle A
ZETA_CYCLE_B	misorientation in the un-recrystallized volume fraction for cycle B
ZETA_CYCLE_C	misorientation in the un-recrystallized volume fraction for cycle C
ZETA_CYCLE_D	misorientation in the un-recrystallized volume fraction for cycle D

5.2.19 Power Law Creep Model

```

BEGIN PARAMETERS FOR MODEL POWER_LAW_CREEP
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU         = <real> 2μ
#
# Viscoplastic parameters
#
CREEP CONSTANT      = <real> A
CREEP EXPONENT      = <real> m
THERMAL CONSTANT    = <real> Q/R
MAX SUBINCREMENTS  = <integer> max_subincrements(100)
END [PARAMETERS FOR MODEL POWER_LAW_CREEP]

```

The power law creep model describes the secondary (or steady-state) creep and is useful in capturing the time-dependent behavior of metals, brazes, or solder at high homologous temperatures. It may also be used as a simple model for the time-dependent behavior of geologic materials such as salt. A general discussion of such creep behaviors and the associated modeling may be found in the texts of [24, 25] while the specific implementation used here is discussed in [5].

In the power law creep model, the effective creep strain rate is taken to be explicitly a function of stress and temperature. A power law relation is used for the stress dependence while an Arrhenius like expression is used to capture thermal effects. As such, the effective creep strain rate is written as,

$$\dot{\bar{\epsilon}}^c = A \bar{\sigma}_{vM}^m \exp\left(\frac{-Q}{R\theta}\right), \quad (5.98)$$

where $\dot{\bar{\epsilon}}^c$ is the effective creep strain rate, $\bar{\sigma}_{vM}$ is the von Mises stress, A is the creep constant, m is the creep exponent, Q is the activation energy, R is the universal gas constant (1.987 cal/mole K), and θ is the absolute temperature. As a slip based mechanism, it is assumed that the creep strains are deviatoric leading to a 3D evolution law of the form,

$$D_{ij}^c = \dot{\bar{\epsilon}}^c \frac{3}{2} \frac{s_{ij}}{\bar{\sigma}_{vM}}, \quad (5.99)$$

with s_{ij} being the deviatoric stress. The corresponding incremental constitutive equation for this model is then given as,

$$\dot{\bar{\sigma}}_{ij} = \mathbb{C}_{ijkl} (D_{kl} - D_{kl}^c). \quad (5.100)$$

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The creep constant, A , in Equation (5.98) is defined with the `CREEP CONSTANT` command line.
- The creep exponent, m , in Equation (5.98) is defined with the `CREEP EXPONENT` command line.
- The thermal constant, Q/R in Equation (5.98) is defined with the `THERMAL CONSTANT` command line.
- time step sub incrementation within the material model may be used to accurately calculate the true creep stress. The maximum sub-increments in a load step is defined with the `MAX SUBINCREMENTS` command line. The default is 100. A larger number of steps can potentially improve accuracy if a large amount of creep happens in a single step. A smaller number of steps can sometimes improve analysis speed.

Output variables available for this model are listed in Table 5.18.

Table 5.18: State Variables for POWER LAW CREEP Model (Section 5.2.19)

Name	Description
ECREEP	equivalent creep strain
SEQDOT	equivalent stress rate

5.2.20 Viscoplastic Model

```

BEGIN PARAMETERS FOR MODEL VISCOPLASTIC
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 

FLOW RATE      = <real>  $g_0$ 
SINH EXPONENT  = <real>  $p_0$ 
ALPHA          = <real>  $\alpha_0$ 
ISO HARDENING  = <real>  $A_1^0$ 
ISO RECOVERY   = <real>  $A_2^0$ 
ISO EXPONENT   = <real>  $A_3$ 
KIN HARDENING  = <real>  $A_4^0$ 
KIN RECOVERY   = <real>  $A_5^0$ 
KIN EXPONENT   = <real>  $A_6$ 
FLOW STRESS    = <real>  $D_0$ 

SHEAR FUNCTION = <string>  $h_G(\theta)$ 
BULK FUNCTION  = <string>  $h_K(\theta)$ 
RATE FUNCTION  = <string>  $h_g(\theta)$ 
EXPONENT FUNCTION = <string>  $h_p(\theta)$ 
ALPHA FUNCTION = <string>  $h_\alpha(\theta)$ 
IHARD FUNCTION = <string>  $h_1(\theta)$ 
IREC FUNCTION  = <string>  $h_2(\theta)$ 
KHARD FUNCTION = <string>  $h_4(\theta)$ 
KREC FUNCTION  = <string>  $h_5(\theta)$ 
MAX SUBINCREMENTS = <int> itmax (2000)
END [PARAMETERS FOR MODEL VISCOPLASTIC]

```

The viscoplastic model is a rate dependent plasticity model that is useful for modeling solders and brazes and was developed by Neilsen et al. [26]. This model is formulated in terms of the stress rate for the material. Like many inelastic models, the rate of deformation, D_{ij} , is additively decomposed into an elastic, D_{ij}^e , and an inelastic, D_{ij}^{in} part such that,

$$D_{ij} = D_{ij}^e + D_{ij}^{\text{in}}. \quad (5.101)$$

The elastic rate of deformation is the only part that contributes to the stress rate and it does so through the elastic moduli, $\mathbb{C}_{ijkl}$, in a linear fashion leading to the relation,

$$\dot{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e, \quad (5.102)$$

where $\mathbb{C}_{ijkl}$ are the components of the fourth-order, isotropic elasticity tensor. The stress rate is arbitrary, as long as it is objective. Two objective stress rates are commonly used: the Jaumann rate and the Green-McInnis rate. For problems with fixed principal axes of deformation, these two rates give the same answers. For problems where the principal axes of deformation rotate during the deformation, the two rates can give different answers. Generally speaking there is no reason to pick one objective rate over another.

The inelastic strain rate is a function of the stress state, σ_{ij} , the temperature, θ , and a number of internal state variables including both scalar isotropic, D , and tensorial kinematic, B_{ij} , hardening variables. With these dependencies defined, a general form for the evolution of the inelastic deformation may be given by,

$$D_{ij}^{\text{in}} = \frac{3}{2} \gamma (\sigma_{ij}, \theta; D, B_{ij}) n_{ij}, \quad (5.103)$$

where n_{ij} is the direction of inelastic deformation and is defined as,

$$n_{ij} = \frac{1}{\tau} \left(s_{ij} - \frac{2}{3} B_{ij} \right), \quad (5.104)$$

and

$$\tau = \sqrt{\frac{3}{2} \left(s_{ij} - \frac{2}{3} B_{ij} \right) \left(s_{ij} - \frac{2}{3} B_{ij} \right)}, \quad (5.105)$$

with s_{ij} being the deviatoric stress tensor. The inelastic strain rate, γ , is defined via a hyperbolic sin law,

$$\gamma = f(\theta) \left[\sinh \left(\frac{\tau}{\alpha(\theta)D} \right) \right]^{p(\theta)}, \quad (5.106)$$

where $f(\theta) = \exp(g(\theta))$. The expressions $g(\theta)$, $\alpha(\theta)$, and $p(\theta)$ are model parameters that are functions of temperature.

The evolution laws for the state variables D and B_{ij} are,

$$\dot{D} = \frac{A_1}{(D - D_0)^{A_3}} \gamma - A_2 (D - D_0)^2, \quad (5.107)$$

and

$$\dot{B}_{ij} = \frac{A_4}{b^{A_6}} D_{ij}^{\text{in}} - A_5 b B_{ij}, \quad (5.108)$$

where

$$b = \sqrt{\frac{2}{3} B_{ij} B_{ij}}. \quad (5.109)$$

The parameters D_0 , A_1 , A_2 , A_3 , A_4 , A_5 and A_6 are model parameters. The parameters A_1 , A_2 , A_4 and A_5 are also functions of temperature. The model can be simplified with the appropriate choice of these parameters.

The following material parameters are functions of temperature and have the following form

$$\begin{aligned} G(\theta) &= G_0 h_G(\theta) \quad ; \quad K(\theta) = K_0 h_K(\theta) \\ g(\theta) &= g_0 h_g(\theta) \quad ; \quad p(\theta) = p_0 h_p(\theta) \quad ; \quad \alpha(\theta) = \alpha_0 h_\alpha(\theta) \\ A_1(\theta) &= A_1^0 h_1(\theta) \quad ; \quad A_2(\theta) = A_2^0 h_2(\theta) \\ A_4(\theta) &= A_4^0 h_4(\theta) \quad ; \quad A_5(\theta) = A_5^0 h_5(\theta) \end{aligned} \quad (5.110)$$

where the functions $h_*(\theta)$ are normalized functions of temperature and the values $(*)_0$ or $(*)^0$ are the reference values that are input in the command block.

In the above command blocks:

- Since the model requires functions to describe the temperature dependence of the bulk and shear modulus, it is recommended that one inputs the bulk and shear modulus at some reference temperature. However, any two of the elastic constants can be used for input. See Section 5.1.5 for more information on elastic constants input.
- The reference value in the equation for the flow rate, $\ln f_0$, is defined with the `FLOW RATE` command line.
- The reference value for the exponent on the sinh function, p_0 , is defined with the `SINH EXPONENT` command line.
- The reference value α_0 is defined with the `ALPHA` command line.
- The reference value for the isotropic hardening parameter, A_1^0 , is defined with the `ISO HARDENING` command line.

- The reference value for the isotropic recovery parameter, A_2^0 , is defined with the `ISO RECOVERY` command line.
- The value for the isotropic hardening exponent parameter, A_3 , is defined with the `ISO EXPONENT` command line.
- The reference value for the kinematic hardening parameter, A_4^0 , is defined with the `KIN HARDENING` command line.
- The reference value for the kinematic recovery parameter, A_5^0 , is defined with the `KIN RECOVERY` command line.
- The value for the kinematic hardening exponent parameter, A_6 , is defined with the `KIN EXPONENT` command line.
- The value for the flow stress, D_0 , is defined with the `FLOW STRESS` command line.
- The user-defined and normalized function that gives the shear modulus as a function of temperature, $h_G(\theta)$, is defined with the `SHEAR FUNCTION` command line.
- The user-defined and normalized function that gives the bulk modulus as a function of temperature, $h_K(\theta)$, is defined with the `BULK FUNCTION` command line.
- The user-defined and normalized function that gives the flow rate as a function of temperature, $h_g(\theta)$, is defined with the `RATE FUNCTION` command line.
- The user-defined and normalized function that gives the sinh exponent as a function of temperature, $h_p(\theta)$, is defined with the `EXPONENT FUNCTION` command line.
- The user-defined and normalized function that gives α as a function of temperature, $h_\alpha(\theta)$, is defined with the `ALPHA FUNCTION` command line.
- The user-defined and normalized function that gives A_1 as a function of temperature, $h_1(\theta)$, is defined with the `IHARD FUNCTION` command line.
- The user-defined and normalized function that gives A_2 as a function of temperature, $h_2(\theta)$, is defined with the `IREC FUNCTION` command line.
- The user-defined and normalized function that gives A_4 as a function of temperature, $h_4(\theta)$, is defined with the `KHARD FUNCTION` command line.
- The user-defined and normalized function that gives A_5 as a function of temperature, $h_5(\theta)$, is defined with the `KREC FUNCTION` command line.
- The Viscoplastic model may need to take sub-increments to solve for the plastic flow over the current time step. The maximum number of steps that may be taken on a step prior to issuing an error can be set by the `MAX SUBINCREMENTS` command line. This value defaults to 2000.

Output variables available for this model are listed in Table 5.19.

More information on the model can be found in the report by Neilsen, et. al. [26].

Table 5.19: State Variables for VISCOPLASTIC Model (Section 5.2.20)

Name	Description
EQPS	equivalent plastic strain
SVB	kinematic hardening variable, $\mathbf{B}$
SVB_XX	kinematic hardening variable - xx component, B_{xx}
SVB_YY	kinematic hardening variable - yy component, B_{yy}
SVB_ZZ	kinematic hardening variable - zz component, B_{zz}
SVB_XY	kinematic hardening variable - xy component, B_{xy}
SVB_YZ	kinematic hardening variable - yz component, B_{yz}
SVB_ZX	kinematic hardening variable - zx component, B_{zx}
SVD	isotropic hardening variable, D
EQDOT	inelastic strain rate, γ
COUNT	number of sub-increments
SHEAR	shear modulus, $G(\theta)$
BULK	bulk modulus, $K(\theta)$
RATE	$g(\theta)$ (see(5.106))
EXP	$p(\theta)$ (see(5.106))
ALPHA	$\alpha(\theta)$ (see(5.106))
A1	isotropic hardening parameter, $A_1(\theta)$
A2	isotropic recovery parameter, $A_2(\theta)$
A4	kinematic hardening parameter, $A_4(\theta)$
A5	kinematic recovery parameter, $A_5(\theta)$

5.2.21 Soil and Crushable Foam Model

```

BEGIN PARAMETERS FOR MODEL SOIL_FOAM
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Yield surface parameters
#
A0 = <real>  $a_0$ 
A1 = <real>  $a_1$ 
A2 = <real>  $a_2$ 
PRESSURE CUTOFF   = <real>  $p_c$ 
PRESSURE FUNCTION = <string>  $f_p(\varepsilon_V)$ 
END [PARAMETERS FOR MODEL SOIL_FOAM]

```

The soil and crushable foam model is a plasticity model that can be used for modeling soil, crushable foam, or other highly compressible materials. Given the right input, the model is a Drucker-Prager model.

For the soil and crushable foam model, the yield surface is a surface of revolution about the hydrostat in principal stress space. A planar end cap is assumed for the yield surface so that the yield surface is closed. The yield stress σ_{yd} is specified as a polynomial in pressure p . The yield stress is given as:

$$\sigma_{yd} = a_0 + a_1 p + a_2 p^2, \quad (5.111)$$

where the pressure p is positive in compression. The determination of the yield stress from Equation (5.111) places severe restrictions on the admissible values of a_0 , a_1 , and a_2 . There are three valid cases for the yield surface. In the first case, there is an elastic–perfectly plastic deviatoric response, and the yield surface is a cylinder oriented along the hydrostat in principal stress space. In this case, a_0 is positive, and a_1 and a_2 are zero. In the second case, the yield surface is conical. A conical yield surface is obtained by setting a_2 to zero and using appropriate values for a_0 and a_1 . In the third case, the yield surface has a parabolic shape. For the parabolic yield surface, all three coefficients in Equation (5.111) are nonzero. The coefficients are checked to determine that a valid negative tensile-failure pressure can be derived based on the specified coefficients.

For the case of the cylindrical yield surface (e.g., $a_0 > 0$ and $a_1 = a_2 = 0$), there is no tensile-failure pressure. For the other two cases, the computed tensile-failure pressure may be too low. To handle the situations where there is no tensile-failure pressure or the tensile-failure pressure is too low,

a pressure cutoff can be defined. If a pressure cutoff is defined, the tensile-failure pressure is the larger of the computed tensile-failure pressure and the defined cutoff pressure.

The plasticity theories for the volumetric and deviatoric parts of the material response are completely uncoupled. The volumetric response is computed first. The mean pressure p is assumed to be positive in compression, and a yield function ϕ_p is written for the volumetric response as:

$$\phi_p = p - f_p(\varepsilon_v) , \quad (5.112)$$

where $f_p(\varepsilon_v)$ defines the volumetric stress-strain curve for the pressure. The yield function ϕ_p determines the motion of the end cap along the hydrostat.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The constant coefficient in the equation for the yield surface (5.111) is defined with the A0 command line.
- The coefficient for the linear term in the equation for the yield surface (5.111) is defined with the A1 command line.
- The coefficient for the quadratic term in the equation for the yield surface (5.111) is defined with the A2 command line.
- The user-defined maximum tensile-failure pressure is defined with the `PRESSURE CUTOFF` command line.
- The pressure as a function of volumetric strain is defined with the function named on the `PRESSURE FUNCTION` command line.

For information about the soil and crushable foam model, see the PRONTO3D document listed as Reference [27]. The soil and crushable foam model is the same as the soil and crushable foam model in PRONTO3D. The PRONTO3D model is based on a material model developed by Krieg [28]. The Krieg version of the soil and crushable foam model was later modified by Swenson and Taylor [29]. The soil and crushable foam model developed by Swenson and Taylor is the model in PRONTO3D and is also the shared model for Presto and Adagio.

Output variables available for this model are listed in Table 5.20.

Table 5.20: State Variables for SOIL FOAM Model (Section [5.2.21](#))

Name	Description
EVOL_MAX	maximum volumetric strain seen by the material point
EVOL_FRAC	volumetric strain for tensile fracture
EVOL	current volumetric strain
EQPS	equivalent plastic strain

5.2.22 Foam Plasticity Model

```
BEGIN PARAMETERS FOR MODEL FOAM_PLASTICITY
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
#
#
PHI             = <real>  $\phi_0$ 
SHEAR STRENGTH = <real>  $A_0$ 
SHEAR HARDENING = <real>  $A_1$ 
SHEAR EXPONENT = <real>  $A_2$ 
HYDRO STRENGTH = <real>  $B_0$ 
HYDRO HARDENING = <real>  $B_1$ 
HYDRO EXPONENT = <real>  $B_2$ 
BETA           = <real>  $\beta$ 
END [PARAMETERS FOR MODEL FOAM_PLASTICITY]
```

The foam plasticity model was developed to describe the response of porous materials (like closed-cell polyurethane foams) exhibiting irreversible, elastic-plastic like responses through large deformations. Such foams can exhibit significant plastic deviatoric and volumetric strains leading to permanent shape and volume changes, respectively. The former behavior is quite typical of metals and corresponding theories are well established. The latter response, however, is not typical of metals and a theory combining these two behaviors is needed. Given these responses of interest, the foam plasticity model is well suited to use with metal foams and many closed-cell polymeric foams (e.g. polyurethane, polystyrene bead, etc.) subjected to large deformations. As permanent strains are of interest, this model is not appropriate for use with flexible foams that return to their undeformed shape after loads are removed.

Specifically, the model developed by Neilsen et al. [30] seeks to capture the response associated with three distinct deformation regimes. First, when foams are initially compressed, they typically exhibit an elastic response. After sufficient load is applied, a plateau of nearly constant stress over a large deformation region is noted as pores start to compress and cell walls undergo substantial deformation. Eventually, the various collapsed cells and walls begin to interact and a densification response with substantial hardening is observed. Details of these deformation processes may be found in the text of Gibson and Ashby [31].

Like other plasticity-based models, the incremental constitutive law for the foam plasticity model is written as,

$$\dot{\sigma}_{ij} = \mathbb{C}_{ijkl} D_{kl}^e, \quad (5.113)$$

where an additive decomposition of the strain rates such that $D_{ij} = D_{ij}^e + D_{ij}^p$ is assumed. To describe the inelastic response of the foams of interest, Neilsen and coworkers [30] proposed a yield function of the form

$$f = \frac{\bar{\sigma}^2}{a^2} + \frac{p^2}{b^2} - 1, \quad (5.114)$$

where $\bar{\sigma}$ is the von Mises effective stress ($\bar{\sigma} = \sqrt{(3/2) s_{ij}s_{ij}}$ with s_{ij} being the deviatoric stress) and p being the hydrostatic pressure that is positive in compression ($p = -(1/3) \sigma_{kk}$). In such a form, the initial yield surface forms an ellipsoid about the hydrostat. The two denominators, a and b , are state variables capturing hardening effects and have the functional form of,

$$a = A_0 + A_1 \phi^{A_2}, \quad (5.115)$$

$$b = \begin{cases} B_0 + B_1 \phi^{B_2} & p \geq 0 \\ B_0 & p < 0 \end{cases}, \quad (5.116)$$

with A_0 , A_1 , A_2 , B_0 , B_1 , and B_2 being model parameters and ϕ being the maximum volume fraction of solid material obtained through the loading history and is defined as,

$$\phi = \phi_0 \frac{V_0}{V}, \quad (5.117)$$

in which ϕ_0 is the initial volume fraction of solid material and V_0 and V are the initial volume and current volume, respectively, of the foam. Put in terms of the deformation,

$$\phi = \phi_0 \frac{1}{1 + \varepsilon_V}, \quad (5.118)$$

where ε_V is the engineering volume strain.

To describe the inelastic plastic deformation, a non-associated flow rule is used. Specifically,

$$D_{ij}^p = \dot{\gamma} g_{ij}, \quad (5.119)$$

where $\dot{\gamma}$ is the consistency multiplier found by enforcing the corresponding condition and

$$g_{ij} = (1 - \beta) g_{ij}^a + \beta g_{ij}^r, \quad (5.120)$$

with the superscripts “a” and “r” being used to denote associated and radial flow directions, respectively. The model parameter β is introduced in (5.120) to enable associated ($\beta = 0$), radial ($\beta = 1$),

or a linear combination of the two flow rules ($0 < \beta < 1$) to be used. The two direction vectors may be written as,

$$g_{ij}^a = \frac{\frac{\partial f}{\partial \sigma_{ij}}}{\left| \frac{\partial f}{\partial \sigma_{kl}} \right|} = \frac{\frac{3}{a^2} s_{ij} - \frac{2}{3b^2} p \delta_{ij}}{\left| \frac{3}{a^2} s_{ij} - \frac{2}{3b^2} p \delta_{ij} \right|}, \quad (5.121)$$

$$g_{ij}^r = \frac{\sigma_{ij}}{|\sigma_{kl}|} = \frac{\sigma_{ij}}{\sqrt{\sigma_{kl} \sigma_{kl}}}. \quad (5.122)$$

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input.
- The initial volume fraction of solid material in the foam, φ , is defined with the `PHI` command line. For example, solid polyurethane weighs 75 pounds per cubic foot (pcf); uncompressed 10 pcf polyurethane foam would have a φ of $0.133 = 10/75$.
- The shear (deviatoric) strength of uncompressed foam is defined with the `SHEAR STRENGTH` command line.
- The shear hardening modulus for the foam is defined with the `SHEAR HARDENING` command line.
- The shear hardening exponent is defined with the `SHEAR EXPONENT` command line. The deviatoric strength is given by `(SHEAR STRENGTH) + (SHEAR HARDENING) * PHI ** (SHEAR EXPONENT)`.
- The hydrostatic (volumetric) strength of the uncompressed foam is defined with the `HYDRO STRENGTH` command line.
- The hydrodynamic hardening modulus is defined with the `HYDRO HARDENING` command line.
- The hydrodynamic hardening exponent is defined with the `HYDRO EXPONENT` command line. The hydrostatic strength is given by `(HYDRO STRENGTH) + (HYDRO HARDENING) * PHI ** (HYDRO EXPONENT)`.
- The prescription for non associated flow, β , is defined with the `BETA` command line. When $\beta = 0.0$, the flow direction is given by the normal to the yield surface (associated flow). When $\beta = 1.0$, the flow direction is given by the stress tensor. It is recommended that associated flow $\beta = 0.0$ be used for most analyses.

Output variables available for this model are listed in Table 5.21.

Table 5.21: State Variables for FOAM PLASTICITY Model (Section [5.2.22](#))

Name	Description
ITER	iterations
EVOL	volumetric strain
PHI	phi, ϕ
EQPS	equivalent plastic strain, $\bar{\varepsilon}^p$
PA	A
PB	B

5.2.23 Low Density Foam Model

```

BEGIN PARAMETERS FOR MODEL LOW_DENSITY_FOAM
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU         = <real> 2μ
#
A      = <real> A
B      = <real> B
C      = <real> C
NAIR   = <real> Nair
P0     = <real> p0
PHI    = <real> φ
END [PARAMETERS FOR MODEL LOW_DENSITY_FOAM]

```

The low density foam material model is a phenomenological model for rigid, low density polyurethane foams. Development of this model followed extensive characterization efforts at Sandia National Laboratory with special emphasis placed on hydrostatic and triaxial compression tests [32]. A key observation of this investigation was the impact of trapped air inside the foam on the load bearing capabilities of the material.

In constructing a model describing the response of the low-density foams, Neilsen et al. [32] decomposed the response into that of the polymeric skeleton and the air such that,

$$\sigma_{ij} = \sigma_{ij}^{\text{sk}} + \sigma^{\text{air}} \delta_{ij}, \quad (5.123)$$

where the super script “sk” is used to refer to variables relating to the skeleton and “air” to the air. The contribution of the air component is only present, however, in constrained cases when the internal gases are trapped and not allowed to escape. If the foam material is not encased or encapsulated in some way, the air may escape and $\sigma^{\text{air}} = 0$. A model parameter, N_{air} , is included to distinguish between these cases. If N_{air} is set to 0, the air pressure term is set to zero. For any other value, it is included.

Using the ideal gas law, it can be found that for an isothermal case,

$$\sigma^{\text{air}} = \frac{p_0 \varepsilon_V}{\varepsilon_V + 1 - \phi}, \quad (5.124)$$

where p_0 , ε_V , and ϕ are the initial air pressure, volumetric strain, and the volume fraction of the solid (skeleton) material. Knowing the total stress of the material volume and air contribution, the skeleton stress may be found via (5.123). Furthermore, it should be noted that the foam (total) and skeleton strains are the same.

Based on their experimental observations, Neilsen et al. [32] noted a decoupling between the skeleton principal stresses. Therefore, the Poisson's ratio of the skeleton is zero and that the yielding behavior in each principal direction is independent. A yield function of the form,

$$f_i = \sigma_i^{\text{sk}} - \bar{\sigma}, \quad (5.125)$$

where f_i and σ_i^{sk} are the i^{th} yield function and skeleton principal stress, respectively, and

$$\bar{\sigma} = A \langle I_2' \rangle + B(1.0 + C\varepsilon_V) \quad (5.126)$$

with A, B, and C are material parameters, and $\langle \cdot \rangle$ denoting the Heaviside step function where

$$\langle x \rangle = \begin{cases} 0 & \text{if } x \leq 0 \\ 1 & \text{if } x > 0 \end{cases}, \quad (5.127)$$

was proposed. Additionally, I_2' is the second invariant of the deviatoric strain. If a skeleton principal stress indicates yielding, it is set to the effective yield stress value, $\bar{\sigma}$.

See Section 5.1.5 for more information on elastic constants input.

State variables for this model are listed in Table 5.22.

For more information on the low density foam material model, see [32].

Table 5.22: State Variables for LOW DENSITY FOAM Model (Section 5.2.23)

Name	Description
PAIR	Air pressure

5.2.24 Hyperfoam Model

```

BEGIN PARAMETERS FOR MODEL HYPERFOAM
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU         = <real> 2μ
#
# Strain energy density
#
N = <integer> N
SHEAR      = <real_list> μi
ALPHA      = <real_list> αi
POISSON    = <real_list> νi
END [PARAMETERS FOR HYPERFOAM]

```

The hyperfoam model is a hyperelastic model that can be used for modeling elastomeric foams. It is based on a strain energy with a form given by Störakers [33] which is similar to a form presented by Ogden [34]. The strain energy depends on the principal stretch ratios of the material and is given by

$$W(\lambda_k) = \sum_{i=1}^N \frac{2\mu_i}{\alpha_i^2} \left[\lambda_1^{\alpha_i} + \lambda_2^{\alpha_i} + \lambda_3^{\alpha_i} - 3 + \frac{1}{\beta_i} (J^{-\alpha_i \beta_i} - 1) \right] \quad (5.128)$$

where μ_i and α_i are input parameters and J is the determinant of the deformation gradient. The value of β_i is calculated from the parameters ν_i via

$$\beta_i = \frac{\nu_i}{1 - 2\nu_i}. \quad (5.129)$$

The ν_i can be thought of as Poisson's ratios, however in the context of the summation in (5.128) it is best to consider them as fitting parameters.

The strain energy (5.128) is a sum of N contributions. The principal Kirchoff stresses for the hyperfoam model, τ_k , can be calculated as

$$\tau_k = \lambda_k \frac{\partial W}{\partial \lambda_k} \quad (5.130)$$

which can be used to calculate the components of the Kirchoff stress, τ_{ij} , through

$$\tau_{ij} = \sum_{k=1}^3 \tau_k \hat{e}_i^k \hat{e}_j^k. \quad (5.131)$$

where $\hat{e}_i^k$ are the components of the k^{th} eigenvector of the left stretch tensor in the global Cartesian coordinate system. The components of the Cauchy stress are then

$$\sigma_{ij} = \frac{1}{J} \tau_{ij}. \quad (5.132)$$

Finally, it should be noted that the Hyperfoam model is also capable of reproducing the Blatz-Ko model [35, 36]. If only one term is chosen, $N = 1$, and $\mu_1 = \mu$, $\alpha_1 = -2$, and $\nu_1 = 0.25$ we get the Blatz-Ko strain energy density

$$W = \frac{\mu}{2} \left(\frac{I_2}{I_3} + 2 \sqrt{I_3} - 5 \right), \quad (5.133)$$

where I_2 and I_3 are the second and third invariants of the right Cauchy-Green tensor.

- See Section 5.1.5 for more information on elastic constants input.
- The number of terms in the expansion of the strain energy is defined with the `N` command line.
- The shear terms in the expansion of the strain energy, μ_i , are defined with the `SHEAR` command line.
- The alpha terms in the expansion of the strain energy, α_i , are defined with the `ALPHA` command line.
- The Poisson ratio terms in the expansion of the strain energy, ν_i , are defined with the `POISSON` command line.

There are no output variables available for this model.

5.2.25 Elastic Three-Dimensional Orthotropic Model

```

BEGIN PARAMETERS FOR MODEL ELASTIC_3D_ORTHOTROPIC
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Material coordinates system definition
#
COORDINATE SYSTEM           = <string> coordinate_system_name
DIRECTION FOR ROTATION      = <real> 1|2|3
ALPHA                       = <real>  $\alpha_1$  (degrees)
SECOND DIRECTION FOR ROTATION = <real> 1|2|3
SECOND ALPHA                = <real>  $\alpha_2$  (degrees)
#
# Required parameters
#
YOUNGS MODULUS AA = <real>  $E_{AA}$ 
YOUNGS MODULUS BB = <real>  $E_{BB}$ 
YOUNGS MODULUS CC = <real>  $E_{CC}$ 
POISSONS RATIO AB = <real>  $\nu_{AB}$ 
POISSONS RATIO BC = <real>  $\nu_{BC}$ 
POISSONS RATIO CA = <real>  $\nu_{CA}$ 
SHEAR MODULUS AB  = <real>  $G_{AB}$ 
SHEAR MODULUS BC  = <real>  $G_{BC}$ 
SHEAR MODULUS CA  = <real>  $G_{CA}$ 
#
# Thermal strain functions
#
THERMAL STRAIN AA FUNCTION = <string>  $\epsilon_{aa}^{th}(\theta)$ 
THERMAL STRAIN BB FUNCTION = <string>  $\epsilon_{bb}^{th}(\theta)$ 
THERMAL STRAIN CC FUNCTION = <string>  $\epsilon_{cc}^{th}(\theta)$ 
#
END [PARAMETERS FOR MODEL ELASTIC_3D_ORTHOTROPIC]

```

The ELASTIC 3D ORTHOTROPIC model is an extension of the previously discussed ELASTIC routine and describes the linear elastic response of a material which exhibits orthotropic symmetry where the orientation of the principal material directions can be arbitrary. These principal axes are denoted as A, B, and C in the following. Thermal strains are also defined with respect to these principal material axes.

The elastic stiffness for an orthotropic material can be described in terms of the elastic compliance which relates the strain to the stress, $\varepsilon_{ij} = \mathbb{S}_{ijkl}\sigma_{kl}$. For a material with an orthogonal ABC coordinate system, and written in that reference frame, the elastic compliance tensor is given by

$$[\tilde{\mathbb{S}}] = \begin{bmatrix} \frac{1}{E_{AA}} & -\frac{\nu_{BA}}{E_{BB}} & -\frac{\nu_{CA}}{E_{CC}} & 0 & 0 & 0 \\ -\frac{\nu_{AB}}{E_{AA}} & \frac{1}{E_{BB}} & -\frac{\nu_{CB}}{E_{CC}} & 0 & 0 & 0 \\ -\frac{\nu_{AC}}{E_{AA}} & -\frac{\nu_{BC}}{E_{BB}} & \frac{1}{E_{CC}} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{2G_{AB}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{2G_{BC}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{2G_{CA}} \end{bmatrix}, \quad (5.134)$$

where the “ $\tilde{\cdot}$ ” is used to denote a variable in the *ABC* material system.

From the definition (5.134), it can be seen that requiring symmetry leads to relations of the form,

$$\nu_{BA} = \nu_{AB} \frac{E_{BB}}{E_{AA}} \quad ; \quad \nu_{CB} = \nu_{BC} \frac{E_{CC}}{E_{BB}} \quad ; \quad \nu_{AC} = \nu_{CA} \frac{E_{AA}}{E_{CC}}. \quad (5.135)$$

Therefore, only 9 independent constants are needed to fully define the model behavior.

The orthotropic model is also formulated in a hypoelastic fashion, leading to a constitutive equation (in the ABC material frame) of,

$$\overset{\circ}{\tilde{\sigma}}_{ij} = \tilde{\mathbb{C}}_{ijkl} (\tilde{D}_{kl} - \tilde{D}_{kl}^{th}), \quad (5.136)$$

where $\tilde{D}_{ij}^{th}$ is the thermal strain rate.

The elastic stiffness tensor, $\tilde{\mathbb{C}}_{ijkl}$, is the inverse of the compliance, $\tilde{\mathbb{C}}_{ijkl} = \tilde{\mathbb{S}}_{ijkl}^{-1}$, and as such may be determined to be,

$$[\tilde{\mathbb{C}}] = \begin{bmatrix} \mathbb{C}_{AAAA} & \mathbb{C}_{AABB} & \mathbb{C}_{CCAA} & 0 & 0 & 0 \\ \mathbb{C}_{AABB} & \mathbb{C}_{BBBB} & \mathbb{C}_{BBCC} & 0 & 0 & 0 \\ \mathbb{C}_{CCAA} & \mathbb{C}_{BBCC} & \mathbb{C}_{CCCC} & 0 & 0 & 0 \\ 0 & 0 & 0 & 2G_{AB} & 0 & 0 \\ 0 & 0 & 0 & 0 & 2G_{BC} & 0 \\ 0 & 0 & 0 & 0 & 0 & 2G_{CA} \end{bmatrix}. \quad (5.137)$$

where

$$\begin{aligned} \mathbb{C}_{AAAA} &= \frac{1 - \nu_{BC}\nu_{CB}}{\Delta} E_{AA} \quad ; \quad \mathbb{C}_{BBBB} = \frac{1 - \nu_{CA}\nu_{AC}}{\Delta} E_{BB} \quad ; \quad \mathbb{C}_{CCCC} = \frac{1 - \nu_{AB}\nu_{BA}}{\Delta} E_{CC} \\ \mathbb{C}_{AABB} &= \frac{\nu_{BA} + \nu_{CA}\nu_{BC}}{\Delta} E_{AA} \quad ; \quad \mathbb{C}_{BBCC} = \frac{\nu_{CB} + \nu_{AB}\nu_{CA}}{\Delta} E_{BB} \quad ; \quad \mathbb{C}_{CCAA} = \frac{\nu_{AC} + \nu_{BC}\nu_{AB}}{\Delta} E_{CC} \end{aligned} \quad (5.138)$$

and $\Delta = 1 - \nu_{AB}\nu_{BA} - \nu_{BC}\nu_{CB} - \nu_{CA}\nu_{AC} - 2\nu_{AB}\nu_{BC}\nu_{CA}$.

See [37] for more information about the elastic three-dimensional orthotropic model.

In the above command blocks all of the following are required inputs.

- Even though they are not used within the material model itself, elastic constants are still required input for hourglass control, certain preconditioners, and other various capabilities. After examining various test problems, it has been determined that using the mean of the orthotropic properties as the isotropic elastic constants yields the best results. See Section 5.1.5 for more information on elastic constants input.
- See Section 5.1.7 for more information on material coordinates system definition commands.
- The Young's moduli corresponding to the principal material axes A, B, and C are given by the `YOUNGS MODULUS AA`, `YOUNGS MODULUS BB`, and `YOUNGS MODULUS CC` command lines.
- The Poisson's ratio defining the BB normal strain when the material is subjected only to AA normal stress is given by the `POISSONS RATIO AB` command line.
- The Poisson's ratio defining the CC normal strain when the material is subjected only to BB normal stress is given by the `POISSONS RATIO BC` command line.

- The Poisson's ratio defining the AA normal strain when the material is subjected only to CC normal stress is given by the `POISSONS RATIO CA` command line.
- The remaining Poisson's ratios needed for the orthotropic elastic relations (i.e. the BA, CB, and AC Poisson's ratios) are calculated internally. They are calculated as usual from the given Poisson's ratios, given Young's moduli, and energy considerations, which provide expressions for these parameters from the resulting symmetry of the compliance tensor.
- The shear moduli for shear in the AB, BC, and CA planes are given by the `SHEAR MODULUS AB`, `SHEAR MODULUS BC`, and `SHEAR MODULUS CA` command lines, respectively.
- The thermal strain functions for normal thermal strains along the principal material directions are given by the `THERMAL STRAIN AA FUNCTION`, `THERMAL STRAIN BB FUNCTION`, and `THERMAL STRAIN CC FUNCTION` command lines.



Warning: The `ELASTIC_3D_ORTHOTROPIC` model cannot currently be used in conjunction with the control stiffness implicit solver block.

There are no output variables available for the Elastic Three-Dimensional Orthotropic material model.

5.2.26 Elastic Orthotropic Shell Model

```

BEGIN PARAMETERS FOR MODEL ELASTIC_ORTHOTROPIC_SHELL
#
# required parameters
#
YOUNGS MODULUS RR = <real>  $E_{RR}$ 
YOUNGS MODULUS SS = <real>  $E_{SS}$ 
YOUNGS MODULUS TT = <real>  $E_{TT}$ 
POISSONS RATIO RS = <real>  $\nu_{RS}$ 
POISSONS RATIO ST = <real>  $\nu_{ST}$ 
POISSONS RATIO TR = <real>  $\nu_{TR}$ 
SHEAR MODULUS RS = <real>  $G_{RS}$ 
SHEAR MODULUS RT = <real>  $G_{RT}$ 
SHEAR MODULUS ST = <real>  $G_{ST}$ 
END [PARAMETERS FOR ELASTIC_ORTHOTROPIC_SHELL]

```

The ELASTIC ORTHOTROPIC SHELL model describes the linear elastic response of an orthotropic material where the planar orientation of the principal material directions can be arbitrary. This material uses the shell section orthotropic alignment commands described in Section 6.2.4.1 to define the local RST coordinate system.

The general 3D response of an orthotropic material is given above. For the shell model, the ABC coordinate system is replaced with the RST coordinate system. For plane stress the stiffness is calculated with the constraint that $\sigma_{TT} = 0$. From this constraint the thickness strain, which is used to calculate the thickness change for the shell, is

$$\varepsilon_{TT} = -\frac{1}{1 - \nu_{RS}\nu_{SR}} [(\nu_{RT} + \nu_{RS}\nu_{ST})\varepsilon_{RR} + (\nu_{ST} + \nu_{SR}\nu_{RT})\varepsilon_{SS}] \quad (5.139)$$

and results in the following stiffness

$$\begin{Bmatrix} \sigma_{RR} \\ \sigma_{SS} \\ \sigma_{RS} \\ \sigma_{ST} \\ \sigma_{TR} \end{Bmatrix} = \begin{bmatrix} \bar{\mathbb{C}}_{RRRR} & \bar{\mathbb{C}}_{RRSS} & 0 & 0 & 0 \\ \bar{\mathbb{C}}_{RRSS} & \bar{\mathbb{C}}_{SSSS} & 0 & 0 & 0 \\ 0 & 0 & 2G_{RS} & 0 & 0 \\ 0 & 0 & 0 & 2G_{ST} & 0 \\ 0 & 0 & 0 & 0 & 2G_{TR} \end{bmatrix} \begin{Bmatrix} \varepsilon_{RR} \\ \varepsilon_{SS} \\ \varepsilon_{TT} \\ \varepsilon_{RS} \\ \varepsilon_{ST} \\ \varepsilon_{TR} \end{Bmatrix} \quad (5.140)$$

where

$$\bar{\mathbb{C}}_{RRRR} = \frac{E_R}{1 - \nu_{RS}\nu_{SR}} \quad ; \quad \bar{\mathbb{C}}_{SSSS} = \frac{E_S}{1 - \nu_{RS}\nu_{SR}} \quad ; \quad \bar{\mathbb{C}}_{RRSS} = \frac{\nu_{SR}E_R}{1 - \nu_{RS}\nu_{SR}} = \frac{\nu_{RS}E_S}{1 - \nu_{RS}\nu_{SR}} \quad (5.141)$$

In the above command blocks, all of the following are required inputs.

- Young's modulus of the orthogonal R, S, and T axes are defined with the `YOUNGS MODULUS RR`, `YOUNGS MODULUS SS` and `YOUNGS MODULUS TT` command lines.
- `POISSONS RATIO RS` defines the strain in the S direction when the material is pulled in the R direction
- `POISSONS RATIO ST` defines the strain in the T direction when the material is pulled in the S direction
- `POISSONS RATIO TR` defines the strain in the R direction when the material is pulled in the T direction
- The shear moduli in each of shear directions are defined with the `SHEAR MODULUS RS`, `SHEAR MODULUS RT`, and `SHEAR MODULUS ST` command lines.

Error messages for the ELASTIC ORTHOTROPIC SHELL model concern input that results in a non-positive definite stiffness. The error messages, and their meanings, are

Model parameters chosen so that determinant of stiffness < 0

$$1 - \nu_{RS}^2 \frac{E_S}{E_R} - \nu_{ST}^2 \frac{E_T}{E_S} - \nu_{TR}^2 \frac{E_R}{E_T} - 2\nu_{RS}\nu_{ST}\nu_{TR} < 0$$

Model parameters chosen so that RS sub-determinant of stiffness < 0

$$1 - \nu_{RS}^2 \frac{E_S}{E_R} < 0$$

Model parameters chosen so that ST sub-determinant of stiffness < 0

$$1 - \nu_{ST}^2 \frac{E_T}{E_S} < 0$$

Model parameters chosen so that TR sub-determinant of stiffness < 0

$$1 - \nu_{TR}^2 \frac{E_R}{E_T} < 0$$

Warning: In previous releases the `ELASTIC_ORTHOTROPIC_SHELL` model required input parameter `POISSONS RATIO SR`, which could have led to an inconsistent set of parameters. The model also previously did not require `YOUNGS MODULUS TT`, `POISSONS RATIO ST`, or `POISSONS RATIO TR`, which are now required parameters in the current version.



For backward compatibility *only*, the original input syntax remains valid. However, the new behavior of the model is to ignore any input value of ν_{SR} and compute it automatically as $\nu_{SR} = \nu_{RS}(E_{SS}/E_{RR})$. If E_{TT} is not input, it is computed as $(E_{TT} = (E_{RR} + E_{SS})/2)$ by default. If no value is input for ν_{ST} or ν_{TR} , it will default to zero.

For best results *all required values should be input* in future usages of this model.

5.2.27 Elastic Orthotropic Model

```
BEGIN PARAMETERS FOR MODEL ELASTIC_ORTHOTROPIC
#
# Elastic constants
YOUNGS MODULUS = <real>youngs_modulus
POISSONS RATIO = <real>poissons_ratio
SHEAR MODULUS = <real>shear_modulus
BULK MODULUS = <real>bulk_modulus
LAMBDA = <real>lambda
TWO MU = <real>two_mu
#
# required parameters
E11 = <real>e11
E22 = <real>e22
E33 = <real>e33
NU12 = <real>nu12
NU13 = <real>nu13
NU23 = <real>nu23
G12 = <real>g12
G13 = <real>g13
G23 = <real>g23
COORDINATE SYSTEM = <string>coordinate_system_name
#
# optional parameters
ANGLE_1_ABSCISSA = <real>angle_1_abscissa
ANGLE_2_ABSCISSA = <real>angle_2_abscissa
ANGLE_3_ABSCISSA = <real>angle_3_abscissa
ROTATION_AXIS_1 = <real>rotation_axis_1
ROTATION_AXIS_2 = <real>rotation_axis_2
ROTATION_AXIS_3 = <real>rotation_axis_3
ANGLE_1_FUNCTION = <string>angle_1_function_name
ANGLE_2_FUNCTION = <string>angle_2_function_name
ANGLE_3_FUNCTION = <string>angle_3_function_name
THERMAL_STRAIN_11_FUNCTION =
    <string>thermal_strain_11_function_name
THERMAL_STRAIN_22_FUNCTION =
    <string>thermal_strain_22_function_name
THERMAL_STRAIN_33_FUNCTION =
    <string>thermal_strain_33_function_name
END [PARAMETERS FOR MODEL ELASTIC_ORTHOTROPIC]
```

The elastic orthotropic model is a Kirchhoff linear elastic constitutive relation that is useful for modeling polymer matrix composite structures undergoing small strains.

In the above command block, the required inputs are: two of the five general elastic material constants, directional properties, and the coordinate system. The following is a brief description of each input.

- See Section 5.1.5 for more information on elastic constants input.
- The directional moduli E_{11} , E_{22} , and E_{33} are defined with the E11, E22, and E33 command lines.
- The directional Poisson's ratios ν_{12} , ν_{13} , and ν_{23} are defined with the NU12, NU13, and NU23 command lines.
- The directional shear moduli G_{12} , G_{13} , and G_{23} are defined with the G12, G13, and G23 command lines.
- The specification of the principal material directions begins with the selection of a user specified coordinate system given by the COORDINATE SYSTEM command line. Section 2.1.9 describes how to define a user specified coordinate system. See the options below for defining the material orientation.
- Material orientation definition option 1 – Three spatially varying ordered Euler angle functions are given in terms of global coordinates (X, Y, Z = 1, 2, 3) for rotations about and corresponding local axis:
 - The rotation angle function abscissas x_1 , x_2 and x_3 are defined with the ANGLE_1_ABSCISSA, ANGLE_2_ABSCISSA and ANGLE_3_ABSCISSA command lines, respectively. Currently these coordinates pertain to the global system (X, Y, Z) = (1, 2, 3).
 - The axis of rotation i, j and k are defined with the ROTATION_AXIS_1, ROTATION_AXIS_2 and ROTATION_AXIS_3 command lines respectively. The rotation axis is defined for each ordered Euler angle rotation. Thus, each sequential rotation (1 then 2 then 3) defines a new coordinate system in which the subsequent rotation axis and angle are defined.
 - The rotation angle functions $\theta_i(x_1)$, $\theta_j(x_2)$ and $\theta_k(x_3)$ are defined with the ANGLE_1_FUNCTION, ANGLE_2_FUNCTION and ANGLE_3_FUNCTION command lines, respectively. See Section 2.1.5 for defining functions.
- Material orientation definition option 2 – **The above orientation definitions are left undefined or set to zero.** The angles and axes are defined directly to each element integration point using INITIAL CONDITION command blocks, as described in Section 7.3. The angles are defined in the INITIAL CONDITION blocks by using ANGLE_1, ANGLE_2 and ANGLE_3 as the INITIALIZE VARIABLE NAME. The axes are defined by using AXIS_1, AXIS_2 and AXIS_3 as the INITIALIZE VARIABLE NAME.
- Material orientation definition option 3 – **If orientation definitions are left blank and no initial conditions are set,** the rotation tensor is initialized to correspond to the local coordinate system defined in the COORDINATE SYSTEM command line.
- The thermal strain functions for normal thermal strains along the principal material directions are given by the THERMAL_STRAIN_11_FUNCTION, THERMAL_STRAIN_22_FUNCTION, and THERMAL_STRAIN_33_FUNCTION command lines. Currently these are engineering strains. See Section 2.1.5 for defining functions.



Warning: The ELASTIC_ORTHOTROPIC model has not been tested in conjunction with the control stiffness implicit solver block.

The material directions for the `ELASTIC_ORTHOTROPIC` model can be visualized by requesting the element fields `MATERIAL_DIRECTION_1`, `MATERIAL_DIRECTION_2`, and `MATERIAL_DIRECTION_3` in the results output block.

Output variables available for this model are listed in Tables 5.23 and 5.24.

Further questions about the model, or requests for changes/enhancements, can be sent to saengli@sandia.gov.

Table 5.23: State Variables for ELASTIC ORTHOTROPIC Model[1](Section 5.2.27)

Index	Name	Variable Description
1	Q_XX	X component of the material 11 unit vector
2	Q_YY	Y component of the material 22 unit vector
3	Q_ZZ	Z component of the material 33 unit vector
4	Q_XY	Y component of the material 11 unit vector
5	Q_YZ	Z component of the material 22 unit vector
6	Q_ZX	X component of the material 33 unit vector
7	Q_YX	X component of the material 22 unit vector
8	Q_ZY	Y component of the material 33 unit vector
9	Q_XZ	Z component of the material 11 unit vector
10	ANGLE_1	Rotation angle about axis 1 (degrees)
11	ANGLE_2	Rotation angle about axis 2 (degrees)
12	ANGLE_3	Rotation angle about axis 3 (degrees)
13	AXIS_1	First axis of rotation
14	AXIS_2	Second axis of rotation
15	AXIS_3	Third axis of rotation
16	TH_STR_XX	Thermal stretch ratio in material 11 direction
17	TH_STR_YY	Thermal stretch ratio in material 22 direction
18	TH_STR_ZZ	Thermal stretch ratio in material 33 direction
19	MAT_STRAIN_XX	Green Lagrange strain in material 11 direction
20	MAT_STRAIN_YY	Green Lagrange strain in material 22 direction
21	MAT_STRAIN_ZZ	Green Lagrange strain in material 33 direction
22	MAT_STRAIN_XY	Green Lagrange strain in material 12 direction
23	MAT_STRAIN_YZ	Green Lagrange strain in material 23 direction
24	MAT_STRAIN_ZX	Green Lagrange strain in material 31 direction
25	MAT_STRESS_XX	2nd P-K stress in material 11 direction
26	MAT_STRESS_YY	2nd P-K stress in material 22 direction
27	MAT_STRESS_ZZ	2nd P-K stress in material 33 direction
28	MAT_STRESS_XY	2nd P-K stress in material 12 direction
29	MAT_STRESS_YZ	2nd P-K stress in material 23 direction
30	MAT_STRESS_ZX	2nd P-K stress in material 31 direction

Table 5.24: State Variables for ELASTIC ORTHOTROPIC Model [2] (Section [5.2.27](#))

Index	Name	Variable Description
31	MAT_LOG_STRAIN_XX	Log (Hencky) strain in material 11 direction
32	MAT_LOG_STRAIN_YY	Log (Hencky) strain in material 22 direction
33	MAT_LOG_STRAIN_ZZ	Log (Hencky) strain in material 33 direction
34	MAT_LOG_STRAIN_XY	Log (Hencky) strain in material 12 direction
35	MAT_LOG_STRAIN_YZ	Log (Hencky) strain in material 23 direction
36	MAT_LOG_STRAIN_ZX	Log (Hencky) strain in material 31 direction
37	MAT_CAUCHY_STRESS_XX	Cauchy stress in material 11 direction
38	MAT_CAUCHY_STRESS_YY	Cauchy stress in material 22 direction
39	MAT_CAUCHY_STRESS_ZZ	Cauchy stress in material 33 direction
40	MAT_CAUCHY_STRESS_XY	Cauchy stress in material 12 direction
41	MAT_CAUCHY_STRESS_YZ	Cauchy stress in material 23 direction
42	MAT_CAUCHY_STRESS_ZX	Cauchy stress in material 31 direction

5.2.28 Elastic Orthotropic Fail Model

```
BEGIN PARAMETERS FOR MODEL ELASTIC_ORTHOTROPIC_FAIL
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
# Required parameters
#
E11 = <real>e11
E22 = <real>e22
E33 = <real>e33
NU12 = <real>nu12
NU13 = <real>nu13
NU23 = <real>nu23
G12 = <real>g12
G13 = <real>g13
G23 = <real>g23
#
COORDINATE SYSTEM = <string>coordinate_system_name
#
# Normal thresholds
#
TENSILE_MATRIX_STRENGTH_11      = <real>f1mp
COMPRESSIVE_MATRIX_STRENGTH_11 = <real>f1mn
TENSILE_FIBER_STRENGTH_11       = <real>f1fp
COMPRESSIVE_FIBER_STRENGTH_11  = <real>f1fn
TENSILE_MATRIX_STRENGTH_22      = <real>f2mp
COMPRESSIVE_MATRIX_STRENGTH_22 = <real>f2mn
TENSILE_FIBER_STRENGTH_22       = <real>f2fp
COMPRESSIVE_FIBER_STRENGTH_22  = <real>f2fn
TENSILE_MATRIX_STRENGTH_33      = <real>f3mp
COMPRESSIVE_MATRIX_STRENGTH_33 = <real>f3mn
TENSILE_FIBER_STRENGTH_33       = <real>f3fp
COMPRESSIVE_FIBER_STRENGTH_33  = <real>f3fn
#
# Shear thresholds
#
SHEAR_MATRIX_STRENGTH_12        = <real>s12m
SHEAR_FIBER_STRENGTH_12         = <real>s12f
SHEAR_MATRIX_STRENGTH_23        = <real>s23m
SHEAR_FIBER_STRENGTH_23         = <real>s23f
```

```

SHEAR_MATRIX_STRENGTH_13      = <real>s13m
SHEAR_FIBER_STRENGTH_13      = <real>s13f
#
# Fracture parameters
#
TENSILE_FRACTURE_ENERGY_11    = <real>gilp
COMPRESSIVE_FRACTURE_ENERGY_11 = <real>giln
TENSILE_FRACTURE_ENERGY_22    = <real>gi2p
COMPRESSIVE_FRACTURE_ENERGY_22 = <real>gi2n
TENSILE_FRACTURE_ENERGY_33    = <real>gi3p
COMPRESSIVE_FRACTURE_ENERGY_33 = <real>gi3n
SHEAR_FRACTURE_ENERGY_12      = <real>gii12
SHEAR_FRACTURE_ENERGY_23      = <real>gii23
SHEAR_FRACTURE_ENERGY_13      = <real>gii13
CHARACTERISTIC_LENGTH         = <real>l_star
#
# Damage evolution parameters
#
MAXIMUM_COMPRESSIVE_DAMAGE_11  = <real>dmax1n
MAXIMUM_COMPRESSIVE_DAMAGE_22  = <real>dmax2n
MAXIMUM_COMPRESSIVE_DAMAGE_33  = <real>dmax3n
COMPRESSION_COUPLING_FACTOR_11 = <real>a1pn
COMPRESSION_COUPLING_FACTOR_22 = <real>a2pn
COMPRESSION_COUPLING_FACTOR_33 = <real>a3pn
TENSILE_DAMAGE_MODULUS_11      = <real>k1p
COMPRESSIVE_DAMAGE_MODULUS_11  = <real>k1n
TENSILE_DAMAGE_MODULUS_22      = <real>k2p
COMPRESSIVE_DAMAGE_MODULUS_22  = <real>k2n
TENSILE_DAMAGE_MODULUS_33      = <real>k3p
COMPRESSIVE_DAMAGE_MODULUS_33  = <real>k3n
SHEAR_DAMAGE_MODULUS_12        = <real>k12
SHEAR_DAMAGE_MODULUS_23        = <real>k23
SHEAR_DAMAGE_MODULUS_13        = <real>k13
HARDENING_EXPONENT_11          = <real>n11
HARDENING_EXPONENT_22          = <real>n22
HARDENING_EXPONENT_33          = <real>n33
HARDENING_EXPONENT_12          = <real>n12
HARDENING_EXPONENT_23          = <real>n23
HARDENING_EXPONENT_13          = <real>n13
#
# Optional parameters follow
# Orientation Parameters
#
ANGLE_1_ABSCISSA = <real>angle_1_abscissa
ANGLE_2_ABSCISSA = <real>angle_2_abscissa
ANGLE_3_ABSCISSA = <real>angle_3_abscissa
ROTATION_AXIS_1  = <real>rotation_axis_1

```



```

ROTATION_AXIS_2    = <real>rotation_axis_2
ROTATION_AXIS_3    = <real>rotation_axis_3
ANGLE_1_FUNCTION   = <string>angle_1_function_name
ANGLE_2_FUNCTION   = <string>angle_2_function_name
ANGLE_3_FUNCTION   = <string>angle_3_function_name
#
# Coefficient of thermal expansion functions
#
THERMAL_STRAIN_11_FUNCTION = <string>ctel1_function_name
THERMAL_STRAIN_22_FUNCTION = <string>cte22_function_name
THERMAL_STRAIN_33_FUNCTION = <string>cte33_function_name
#
# Temperature dependent property functions
#
E11_FUNCTION       = <string>e11_function_name
E22_FUNCTION       = <string>e22_function_name
E33_FUNCTION       = <string>e33_function_name
NU12_FUNCTION      = <string>nul2_function_name
NU23_FUNCTION      = <string>nu23_function_name
NU13_FUNCTION      = <string>nul3_function_name
G12_FUNCTION       = <string>g12_function_name
G23_FUNCTION       = <string>g23_function_name
G13_FUNCTION       = <string>g13_function_name
#
# Strain rate dependent parameters
#
REFERENCE_STRAIN_RATE          = <real>epsdot0
ELASTIC_RATE_COEFFICIENT_11    = <real>ce11
ELASTIC_RATE_COEFFICIENT_22    = <real>ce22
ELASTIC_RATE_COEFFICIENT_33    = <real>ce33
ELASTIC_RATE_COEFFICIENT_12    = <real>ce12
ELASTIC_RATE_COEFFICIENT_23    = <real>ce23
ELASTIC_RATE_COEFFICIENT_13    = <real>ce13
FIBER_STRENGTH_RATE_COEFFICIENT_11 = <real>cf11
FIBER_STRENGTH_RATE_COEFFICIENT_22 = <real>cf22
FIBER_STRENGTH_RATE_COEFFICIENT_33 = <real>cf33
FIBER_STRENGTH_RATE_COEFFICIENT_12 = <real>cf12
FIBER_STRENGTH_RATE_COEFFICIENT_23 = <real>cf23
FIBER_STRENGTH_RATE_COEFFICIENT_13 = <real>cf13
MATRIX_STRENGTH_RATE_COEFFICIENT_11 = <real>cm11
MATRIX_STRENGTH_RATE_COEFFICIENT_22 = <real>cm22
MATRIX_STRENGTH_RATE_COEFFICIENT_33 = <real>cm33
MATRIX_STRENGTH_RATE_COEFFICIENT_12 = <real>cm12
MATRIX_STRENGTH_RATE_COEFFICIENT_23 = <real>cm23
MATRIX_STRENGTH_RATE_COEFFICIENT_13 = <real>cm13
END [PARAMETERS FOR MODEL ELASTIC_ORTHOTROPIC_FAIL]

```

The elastic orthotropic fail model is an empirically based constitutive relation that is useful for modeling polymer matrix composite structures. Refer to the SAND report by English [38] for a full description of the material model theory and usage.

In the above command block, the required inputs are: two of the five general elastic material constants, directional properties, and the coordinate system. This model has all of the inputs used in the Elastic Orthotropic Model detailed in Section 5.2.27. The following is a brief description of the additional inputs for the Elastic Orthotropic Fail Model.

- The strengths for each component of damage are given by the commands:

```
# Normal thresholds
TENSILE_MATRIX_STRENGTH_11      = <real>f1mp
COMPRESSIVE_MATRIX_STRENGTH_11 = <real>f1mn
TENSILE_FIBER_STRENGTH_11       = <real>f1fp
COMPRESSIVE_FIBER_STRENGTH_11  = <real>f1fn
TENSILE_MATRIX_STRENGTH_22      = <real>f2mp
COMPRESSIVE_MATRIX_STRENGTH_22 = <real>f2mn
TENSILE_FIBER_STRENGTH_22       = <real>f2fp
COMPRESSIVE_FIBER_STRENGTH_22  = <real>f2fn
TENSILE_MATRIX_STRENGTH_33      = <real>f3mp
COMPRESSIVE_MATRIX_STRENGTH_33 = <real>f3mn
TENSILE_FIBER_STRENGTH_33       = <real>f3fp
COMPRESSIVE_FIBER_STRENGTH_33  = <real>f3fn

# Shear thresholds
SHEAR_MATRIX_STRENGTH_12        = <real>s12m
SHEAR_FIBER_STRENGTH_12         = <real>s12f
SHEAR_MATRIX_STRENGTH_23        = <real>s23m
SHEAR_FIBER_STRENGTH_23         = <real>s23f
SHEAR_MATRIX_STRENGTH_13        = <real>s13m
SHEAR_FIBER_STRENGTH_13         = <real>s13f
```

The terms f1mp, f1mn, f1fp, f1fn, f2mp, f2mn, f2fp, f2fn, f3mp, f3mn, f3fp, f3fn, s12m, s12f, s23m, s23f, s13m and s13f can be interpreted by the following naming convention. The first letter indicates a normal (f) or shear (s) strength. The numbers correspond to the orthotropic planes of damage evolution. The next letter indicates matrix (m) or fiber (f) damage. The matrix mode corresponds with hardening-like behavior and the fiber mode is softening. For normal strength components, the last letter indicates tensile (p for positive) or compressive (n for negative).

- The fracture energies (energy per unit area) for each plane of damage are given by the commands:

```
# Fracture parameters
TENSILE_FRACTURE_ENERGY_11      = <real>gilp
COMPRESSIVE_FRACTURE_ENERGY_11 = <real>giln
TENSILE_FRACTURE_ENERGY_22      = <real>gi2p
COMPRESSIVE_FRACTURE_ENERGY_22 = <real>gi2n
TENSILE_FRACTURE_ENERGY_33      = <real>gi3p
COMPRESSIVE_FRACTURE_ENERGY_33 = <real>gi3n
SHEAR_FRACTURE_ENERGY_12        = <real>gii12
SHEAR_FRACTURE_ENERGY_23        = <real>gii23
```

```

SHEAR_FRACTURE_ENERGY_13      = <real>gii13
CHARACTERISTIC_LENGTH          = <real>l_star

```

The terms $gi1p$, $gi1n$, $gi2p$, $gi2n$, $gi3p$, $gi3n$, $gii12$, $gii23$ and $gii13$ can be interpreted by the following naming convention. For normal components, the first two letters indicate the term is a normal fracture energy (gi), the number indicates the plane of damage and the last letter indicates tensile (p for positive) or compressive (n for negative). For shear components, the first three letters indicate the term is a shear fracture energy (gii) and the number indicates the shear direction. The total energy density dissipated (area under stress-strain curve) is the fracture energy divided by the characteristic length l_star .

- The maximum allowable normal damage values under compression are given by the commands:

```

MAXIMUM_COMPRESSIVE_DAMAGE_11 = <real>dmax1n
MAXIMUM_COMPRESSIVE_DAMAGE_22 = <real>dmax2n
MAXIMUM_COMPRESSIVE_DAMAGE_33 = <real>dmax3n

```

The terms $dmax1n$, $dmax2n$ and $dmax3n$ denote the maximum allowable compressive damage $dmax$ for each plane of damage.

- The proportion of tensile damage translating to compressive damage for each of the orthotropic planes are given by the commands:

```

COMPRESSION_COUPLING_FACTOR_11 = <real>a1pn
COMPRESSION_COUPLING_FACTOR_22 = <real>a2pn
COMPRESSION_COUPLING_FACTOR_33 = <real>a3pn

```

The terms $a1pn$, $a2pn$ and $a3pn$ denote this proportion a of the tensile-to-compressive damage (pn for positive to negative directions) for each plane.

- The slopes of the matrix mode damaging portion of the stress-strain curve are given by the commands:

```

TENSILE_DAMAGE_MODULUS_11      = <real>k1p
COMPRESSIVE_DAMAGE_MODULUS_11  = <real>k1n
TENSILE_DAMAGE_MODULUS_22      = <real>k2p
COMPRESSIVE_DAMAGE_MODULUS_22  = <real>k2n
TENSILE_DAMAGE_MODULUS_33      = <real>k3p
COMPRESSIVE_DAMAGE_MODULUS_33  = <real>k3n
SHEAR_DAMAGE_MODULUS_12        = <real>k12
SHEAR_DAMAGE_MODULUS_23        = <real>k23
SHEAR_DAMAGE_MODULUS_13        = <real>k13

```

For the terms $k1p$, $k1n$, $k2p$, $k2n$, $k3p$, $k3n$, $k12$, $k23$ and $k13$, the k indicates a damage modulus term, the number indicates the direction with respect to the damage planes, and the letters p or n indicate tensile or compressive directions, respectively.

- Small nonlinearity in the matrix mode damage evolution can be added using the hardening exponents given for each of the orthotropic planes, by the commands:

```

HARDENING_EXPONENT_11          = <real>n11
HARDENING_EXPONENT_22          = <real>n22
HARDENING_EXPONENT_33          = <real>n33
HARDENING_EXPONENT_12          = <real>n12
HARDENING_EXPONENT_23          = <real>n23
HARDENING_EXPONENT_13          = <real>n13

```

For the terms $n11$, $n22$, $n33$, $n12$, $n23$ and $n13$, the n indicates a hardening exponent term

and the number indicates the direction.

- Strain rate dependence is defined by the commands:

```
REFERENCE_STRAIN_RATE           = <real>epsdot0
ELASTIC_RATE_COEFFICIENT_11     = <real>ce11
ELASTIC_RATE_COEFFICIENT_22     = <real>ce22
ELASTIC_RATE_COEFFICIENT_33     = <real>ce33
ELASTIC_RATE_COEFFICIENT_12     = <real>ce12
ELASTIC_RATE_COEFFICIENT_23     = <real>ce23
ELASTIC_RATE_COEFFICIENT_13     = <real>ce13
FIBER_STRENGTH_RATE_COEFFICIENT_11 = <real>cf11
FIBER_STRENGTH_RATE_COEFFICIENT_22 = <real>cf22
FIBER_STRENGTH_RATE_COEFFICIENT_33 = <real>cf33
FIBER_STRENGTH_RATE_COEFFICIENT_12 = <real>cf12
FIBER_STRENGTH_RATE_COEFFICIENT_23 = <real>cf23
FIBER_STRENGTH_RATE_COEFFICIENT_13 = <real>cf13
MATRIX_STRENGTH_RATE_COEFFICIENT_11 = <real>cm11
MATRIX_STRENGTH_RATE_COEFFICIENT_22 = <real>cm22
MATRIX_STRENGTH_RATE_COEFFICIENT_33 = <real>cm33
MATRIX_STRENGTH_RATE_COEFFICIENT_12 = <real>cm12
MATRIX_STRENGTH_RATE_COEFFICIENT_23 = <real>cm23
MATRIX_STRENGTH_RATE_COEFFICIENT_13 = <real>cm13
```

The rate dependence is calculated with respect to the reference strain rate `epsdot0`. The rate coefficients for the purely empirical rate equation are given for elastic moduli and failure parameters by the scalar values `ce11`, `ce22`, `ce33`, `ce12`, `ce23`, `ce13`, `cf11`, `cf22`, `cf33`, `cf12`, `cf23`, `cf13`, `cm11`, `cm22`, `cm33`, `cm12`, `cm23` and `cm13`. The letter *c* indicates a rate coefficient, *e* indicates an elastic coefficient, *f* indicates a fiber strength coefficient, *m* indicates matrix strength coefficient, and the number indicates the direction.

The material directions for the `ELASTIC_ORTHOTROPIC_FAIL` model can be visualized by requesting the element fields `MATERIAL_DIRECTION_1`, `MATERIAL_DIRECTION_2`, and `MATERIAL_DIRECTION_3` in the results output block.



Warning: The `ELASTIC_ORTHOTROPIC_FAIL` model has not been tested in conjunction with the control stiffness implicit solver block.

Output variables available for this model are listed in Tables [5.23](#), [5.24](#) and [5.25](#).

Further questions about the model, or requests for changes/enhancements, can be sent to saengli@sandia.gov.

5.2.28.1 Warnings and Usage Guidelines



Warning: Strongly rate-dependent models may fare poorly in implicit quasistatic solution. In implicit analyses the rate term used to evaluate the current load step is the rate seen by the model in the previous load step. This may cause the solution to oscillate between high- and low-rate equilibrium states from step to step.

Table 5.25: Additional State Variables for ELASTIC ORTHOTROPIC FAIL Model (Section 5.2.28)

Index	Name	Variable Description
43	R1MP	Damage evolution variable 11, matrix, tension
44	R1FP	Damage evolution variable 11, fiber, tension
45	R1MN	Damage evolution variable 11, matrix, compression
46	R1FN	Damage evolution variable 11, fiber, compression
47	R2MP	Damage evolution variable 22, matrix, tension
48	R2FP	Damage evolution variable 22, fiber, tension
49	R2MN	Damage evolution variable 22, matrix, compression
50	R2FN	Damage evolution variable 22, fiber, compression
51	R3MP	Damage evolution variable 33, matrix, tension
52	R3FP	Damage evolution variable 33, fiber, tension
53	R3MN	Damage evolution variable 33, matrix, compression
54	R3FN	Damage evolution variable 33, fiber, compression
55	D1MP	Normal damage 11, matrix, tension
56	D1FP	Normal damage 11, fiber, tension
57	D1MN	Normal damage 11, matrix, compression
58	D1FN	Normal damage 11, fiber, compression
59	D2MP	Normal damage 22, matrix, tension
60	D2FP	Normal damage 22, fiber, tension
61	D2MN	Normal damage 22, matrix, compression
62	D2FN	Normal damage 22, fiber, compression
63	D3MP	Normal damage 33, matrix, tension
64	D3FP	Normal damage 33, fiber, tension
65	D3MN	Normal damage 33, matrix, compression
66	D3FN	Normal damage 33, fiber, compression
67	D12M	Shear damage 12, matrix
68	D12F	Shear damage 12, fiber
69	D23M	Shear damage 23, matrix
70	D23F	Shear damage 23, fiber
71	D13M	Shear damage 13, matrix
72	D13F	Shear damage 13, fiber
73	ORTHOTROPIC_DAMAGE _XX	Effective and active normal damage 11
74	ORTHOTROPIC_DAMAGE _YY	Effective and active normal damage 22
75	ORTHOTROPIC_DAMAGE _ZZ	Effective and active normal damage 33
76	ORTHOTROPIC_DAMAGE _XY	Effective and active shear damage 12
77	ORTHOTROPIC_DAMAGE _YZ	Effective and active shear damage 23
78	ORTHOTROPIC_DAMAGE _ZX	Effective and active shear damage 31

5.2.29 Wire Mesh Model

```

BEGIN PARAMETERS FOR MODEL WIRE_MESH
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
TWO MU         = <real> 2μ
#
# Yield surface parameters
#
YIELD FUNCTION = <string>  $\bar{\sigma}(\epsilon_V)$ 
TENSION        = <real>  $\tau$ 
END [PARAMETERS FOR MODEL WIRE_MESH]

```

The wire mesh model was developed at Sandia National Laboratories for use with layered sequences of metallic wire meshes and cloth fabric. Model development was based on an extensive series of experiments performed on these materials (see [39]) and used an existing model for rigid polyurethane foams as a starting point [32].

To be able to analyze the response of this material, the Cauchy stress tensor is first decomposed into its principal components, σ^i . Each principal stress is evaluated independently and two behaviors are considered depending on whether or not the material is in tension or compression. Under a tensile load, the material is taken to be perfectly plastic above a yield stress, τ . For compressive loads, it is assumed that the materials hardens functionally with the volumetric engineering strain, ϵ_V . In this formulation, an arbitrary form of this hardening function, $\bar{\sigma}(\epsilon_V)$ is assumed although in the original work [39],

$$\bar{\sigma}(\epsilon_V) = ae^{-b\epsilon_V}, \quad (5.142)$$

with a and b as material constants, was used.

With these assumptions, the yield function of the i^{th} principal stress, f^i , may be written as,

$$f^i = \begin{cases} \sigma^i - \tau, & \sigma^i \geq 0 \\ -\sigma^i - \bar{\sigma}(\epsilon_V), & \sigma^i < 0 \end{cases} \quad (5.143)$$

where τ is the isotropic tensile strength of the material.

Similar to the rigid polyurethane foam model [39], the flow rule is defined as:

$$d_{ij}^p = \dot{\gamma}^1 P_{ijkl}^1 \sigma_{kl} + \dot{\gamma}^2 P_{ijkl}^2 \sigma_{kl} + \dot{\gamma}^3 P_{ijkl}^3 \sigma_{kl} \quad (5.144)$$

with $\dot{\gamma}^i$ being the magnitude of the i^{th} plastic strain increment and P_{ijkl}^r is the fourth-order principal projection operator defined as,

$$P_{ijkl}^r = n_i^r n_j^r n_k^r n_l^r \quad (5.145)$$

in which n_i^r is the corresponding direction vector of principal stress, σ^r . With this definition,

$$\sigma^r = \sigma_{ij} P_{ijkl}^r \sigma_{kl}. \quad (5.146)$$

- See Section 5.1.5 for more information on elastic constants input.
- The yield function in compression is defined with the `YIELD FUNCTION` command line.
- The tensile strength is give by the `TENSION` command line.

Output variables available for this model are listed in Table 5.26.

More information on the model can be found in the report by Neilsen, et. al. [39].

Table 5.26: State Variables for WIRE MESH Model (Section 5.2.29)

Name	Description
EVOL	engineering volumetric strain
YIELD	current yield strength in compression

5.2.30 Orthotropic Crush Model

```

BEGIN PARAMETERS FOR MODEL ORTHOTROPIC_CRUSH
#
# Elastic constants - Post lock-up
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Orthotropic Elastic properties - Pre-Crush
#
EX              = <real>  $E_{11}$ 
EY              = <real>  $E_{22}$ 
EZ              = <real>  $E_{33}$ 
GXY             = <real>  $G_{12}$ 
GYZ             = <real>  $G_{23}$ 
GZX             = <real>  $G_{31}$ 
#
# Crush properties
#
CRUSH XX        = <string>  $\hat{\sigma}_{11}(J_c)$ 
CRUSH YY        = <string>  $\hat{\sigma}_{22}(J_c)$ 
CRUSH ZZ        = <string>  $\hat{\sigma}_{33}(J_c)$ 
CRUSH XY        = <string>  $\hat{\sigma}_{12}(J_c)$ 
CRUSH YZ        = <string>  $\hat{\sigma}_{23}(J_c)$ 
CRUSH ZX        = <string>  $\hat{\sigma}_{31}(J_c)$ 
VMIN            = <real>  $V_{min}$ 
#
# Post lock-up yield properties
#
YIELD STRESS    = <real>  $\sigma_y$ 
#
END [PARAMETERS FOR MODEL ORTHOTROPIC_CRUSH]

```

The orthotropic crush model in LAMÉ is designed to model the energy absorbing capability of crushable orthotropic materials, e.g. aluminum honeycomb, and is empirically based. The formulation follows that used for metallic honeycomb materials in LS-DYNA [40]. Three response regimes are assumed for this material: (i) orthotropic elastic, (ii) crush, and (iii) complete compaction (fully crushed). During the elastic regime, the model exhibits the response of an elastic, orthotropic material with *all Poisson's ratio equal to zero*. After full compaction, the response is taken to be that of an isotropic, perfectly plastic material and the response between these two stages is tailored to smoothly transition between the two extremes. Crushing, incorporating both nonlinear elastic and plastic-like behaviors, is taken to begin as soon as volumetric contraction is noted ($J = \det(F_{ij}) < 1$). As such, the purely elastic response is primarily seen during cyclic

loadings in which the material is unloaded. An internal state variable, J_c , is introduced to track the crushed state of the material and is defined as the minimum J over the entire deformation history such that,

$$J_c = \min_{t>0} [J(t)]. \quad (5.147)$$

The crushing process manifests through two distinct behaviors: (i) the elastic properties scale linearly with the crush state from the initial orthotropic state to the of the final isotropic completely compacted material; and (ii) a plastic-like response is observed associated with corresponding crush curves (analogous to hardening curves).

Before complete compaction, the incremental constitutive relation may be written in terms of the rate of deformation tensor, D_{ij} , as,

$$\begin{Bmatrix} \dot{\sigma}_{11} \\ \dot{\sigma}_{22} \\ \dot{\sigma}_{33} \\ \dot{\sigma}_{12} \\ \dot{\sigma}_{23} \\ \dot{\sigma}_{31} \end{Bmatrix} = \begin{bmatrix} \hat{E}_{11} & 0 & 0 & 0 & 0 & 0 \\ 0 & \hat{E}_{22} & 0 & 0 & 0 & 0 \\ 0 & 0 & \hat{E}_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & 2\hat{G}_{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & 2\hat{G}_{23} & 0 \\ 0 & 0 & 0 & 0 & 0 & 2\hat{G}_{31} \end{bmatrix} \begin{Bmatrix} D_{11} \\ D_{22} \\ D_{33} \\ D_{12} \\ D_{23} \\ D_{31} \end{Bmatrix} \quad (5.148)$$

where $\hat{E}_{11}$, $\hat{E}_{22}$, and $\hat{E}_{33}$ are the normal stiffness and $\hat{G}_{12}$, $\hat{G}_{23}$, and $\hat{G}_{31}$ are the shear stiffness. A clear decoupling between the different directional components is evident in (5.148). All six stiffness components are assumed to be functions of the current compaction level which may be defined as $1 - J_c$ and the evolution of these terms is responsible for crushing behavior (i) alluded to previously.

The functional forms of the stiffness are given by,

$$\begin{aligned} \hat{E}_{\beta} &= E_{\beta} + \alpha (E - E_{\beta}) & \beta &= 11, 22, 33 \\ \hat{G}_{\gamma} &= G_{\gamma} + \alpha (G - G_{\gamma}) & \gamma &= 12, 23, 31, \end{aligned} \quad (5.149)$$

where E and G are the Young's and shear moduli, respectively, of the fully compacted material while E_{β} and G_{γ} are the input orthotropic elastic stiffness components of the virgin, uncompacted material. It is assumed that these stiffness vary linearly between the pre- and post-compacted material such that,

$$\alpha = \frac{(1 - J_c)}{V_{min}}, \quad (5.150)$$

with V_{min} being the minimum relative volume (or maximum compaction).

With respect to the second behavior observed during crushing, a plastic-like response governed by crush curves is observed. Given the decoupling between the different stresses and deformations, a crush curve needs to be defined for each of the six normal and shear stresses. An example of such a curve is presented in Figure 5.10, and three distinct regions are evident. Initially, at low compaction levels, a plateau is observed. This plateau is essentially an initial crush strength and prior to this stress level all nonlinear deformations associated with material compaction manifest through changes in the respective moduli. When the stress reaches the specified levels, however, the curves play a role analogous to the hardening curve and the material stress follows the curve. Physically, the plateau is associated with crushing the internal honeycomb or foam structure of the material. As the material approaches full compaction and microstructural contact effects become important, a sharp rise in the stress is noted (see $\approx 0.6 \leq 1 - J_c \leq 0.7 = V_{min}$ in Figure 5.10). After complete compaction another plateau corresponding to perfect plasticity is evident.

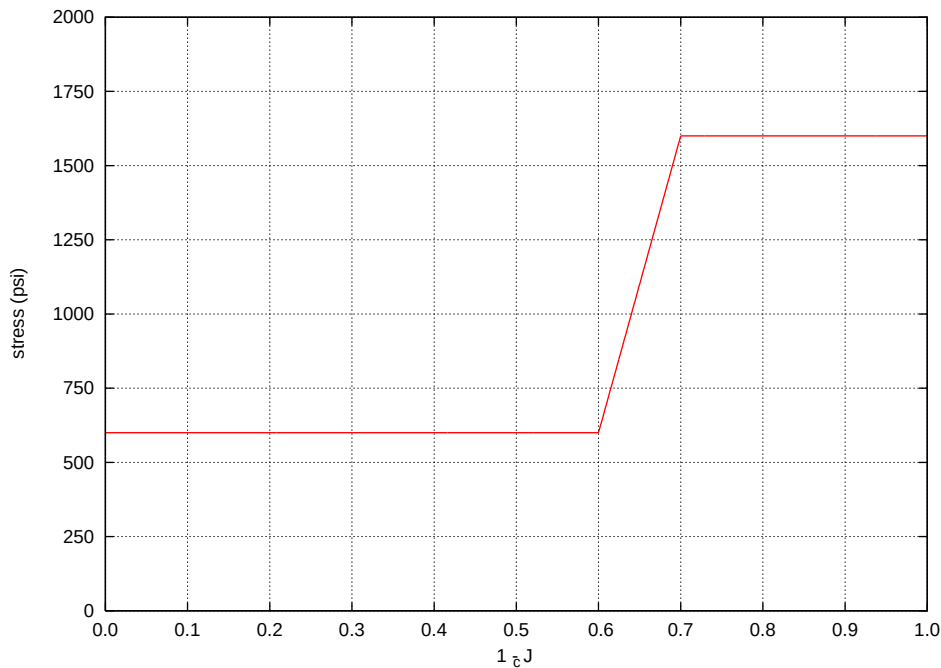


Figure 5.10: An example of an input crush curve for an aluminum honeycomb.

Above some value of compaction ($1 - J_c = V_{min}$), the material will be fully compacted and behave as an elastic, perfectly plastic material. The fully compacted response is given by the Young's modulus, E , Poisson's ratio, ν , and the yield stress, σ_y . Details of this response may be found in previous sections on the various elastic-plastic models (e.g. Section 5.2.5).

In the above command blocks:

- The EX, EY, EZ, GXY, GYZ, and GZX command lines define, respectively, the initial, pre-crush directional moduli E_{xx} , E_{yy} , E_{zz} , G_{xy} , G_{yz} , and G_{zx} from (5.148).
- CRUSH XX, YY, ZZ, XY, YZ, and ZX inputs require the name of a function defined via a FUNCTION command line in the SIERRA scope. These functions describe the directional

crush characteristics of the material and give the current stress value (in a direction) as a function of the current compaction ($1 - J_c$).

- The command `VMIN` defines the minimum relative volume of the material that is achieved when the material is completely crushed. This parameter may also be considered as the maximum compaction.
- The elastic constant commands refer to the *post lock-up, fully compacted* isotropic response of the material. See Section 5.1.5 for more information on elastic constants input.
- `YIELD STRESS` refers to the plateau stress of the material after lock-up when the response is perfectly plastic.

Output variables available for this model are listed in Table 5.27. For information about the orthotropic crush model, consult [40].

Table 5.27: State Variables for ORTHOTROPIC CRUSH Model (Section 5.2.30)

Name	Description
CRUSH	current (unrecoverable) compaction/relative volume

5.2.31 Orthotropic Rate Model

```

BEGIN PARAMETERS FOR MODEL ORTHOTROPIC_RATE
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
YIELD STRESS = <real>  $\sigma_y$ 
#
MODULUS TTTT = <real>  $E_{TTTT}^0$ 
MODULUS TTLL = <real>  $E_{TTLL}^0$ 
MODULUS TTWW = <real>  $E_{TTWW}^0$ 
MODULUS LLLL = <real>  $E_{LLLL}^0$ 
MODULUS LLWW = <real>  $E_{LLWW}^0$ 
MODULUS WWWW = <real>  $E_{WWWW}^0$ 
MODULUS TLTL = <real>  $G_{TLTL}^0$ 
MODULUS LWLW = <real>  $G_{LWLW}^0$ 
MODULUS WTWT = <real>  $G_{WTWT}^0$ 
#
TX = <real>  $\hat{i}_x$ 
TY = <real>  $\hat{i}_y$ 
TZ = <real>  $\hat{i}_z$ 
LX = <real>  $\hat{l}_x$ 
LY = <real>  $\hat{l}_y$ 
LZ = <real>  $\hat{l}_z$ 
#
MODULUS FUNCTION = <string>  $f_E(\alpha)$ 
RATE FUNCTION     = <string>  $h(\dot{\epsilon})$ 
#
T FUNCTION = <string>  $f_{TT}(\alpha)$ 
L FUNCTION = <string>  $f_{LL}(\alpha)$ 
W FUNCTION = <string>  $f_{WW}(\alpha)$ 
TL FUNCTION = <string>  $f_{TL}(\alpha)$ 
LW FUNCTION = <string>  $f_{LW}(\alpha)$ 
WT FUNCTION = <string>  $f_{WT}(\alpha)$ 
END [PARAMETERS FOR MODEL ORTHOTROPIC_RATE]

```

The orthotropic rate model extends the functionality of the orthotropic crush constitutive model described in Section 5.2.30. The orthotropic rate model has been developed to describe the behavior of an aluminum honeycomb subjected to large deformation. The orthotropic rate model, like the original orthotropic crush model, has six independent yield functions that evolve with volume strain. Unlike the orthotropic crush model, the orthotropic rate model has yield functions that also

depend on strain rate. The orthotropic rate model also uses an orthotropic elasticity tensor with nine elastic moduli in place of the orthotropic elasticity tensor with six elastic moduli used in the orthotropic crush model. A honeycomb orientation capability is included with the orthotropic rate model that allows users to prescribe initial honeycomb orientations that are not aligned with the original global coordinate system.

In the above command blocks:

- For the `ORTHOTROPIC_RATE` model, only `YOUNGS MODULUS` needs to be defined; this is the Young's modulus in the fully compacted state. If two other elastic constants are provided, they will be used to define the fully compacted Young's modulus. See Section 5.1.5 for more information on elastic constants input.
- The yield stress for the fully compacted honeycomb is defined with the `YIELD STRESS` command line.
- The nine elastic moduli for the orthotropic, uncompacted honeycomb are defined with the `MODULUS TTT`, `MODULUS TTLL`, `MODULUS TTWW`, `MODULUS LLLL`, `MODULUS LLWW`, `MODULUS WWWW`, `MODULUS TLTL`, `MODULUS LWLW`, and `MODULUS WTWT` command lines. The T-direction is usually associated with the generator axis for the honeycomb. The L-direction is in the ribbon plane (plane defined by flat sheets used in reinforced honeycomb) and orthogonal to the generator axis. The W-direction is perpendicular to the ribbon plane.
- The components of a vector defining the T-direction of the honeycomb are defined by the `TX`, `TY`, and `TZ` command lines. The values of `tx`, `ty`, and `tz` are components of a vector in the global coordinate system that define the orientation of the honeycomb's T-direction (generator axis).
- The components of a vector defining the L-direction of the honeycomb are defined by the `LX`, `LY`, and `LZ` command lines. The values of `lx`, `ly`, and `lz` are components of a vector in the global coordinate system that define the orientation of the honeycomb's L-direction. *Caution:* The vectors T and L must be orthogonal.
- The function describing the variation in moduli with compaction is given by the `MODULUS FUNCTION` command line. The moduli vary continuously from their initial orthotropic values to isotropic values when full compaction is obtained.
- The function describing the change in strength with strain rate is given by the `RATE FUNCTION` command line. Note that all strengths are scaled with the multiplier obtained from this function.
- The function describing the T-normal strength of the honeycomb as a function of compressive volumetric strain is given by the `T FUNCTION` command line.
- The function describing the L-normal strength of the honeycomb as a function of compressive volumetric strain is given by the `L FUNCTION` command line.
- The function describing the W-normal strength of the honeycomb as a function of compressive volumetric strain is given by the `W FUNCTION` command line.

- The function describing the TL-normal strength of the honeycomb as a function of compressive volumetric strain is given by the `TL FUNCTION` command line.
- The function describing the LW-normal strength of the honeycomb as a function of compressive volumetric strain is given by the `LW FUNCTION` command line.
- The function describing the WT-normal strength of the honeycomb as a function of compressive volumetric strain is given by the `WT FUNCTION` command line.

Note that several of the command lines in this command block reference functions. These functions must be defined in the SIERRA scope.

Output variables for this model are listed in Table 5.28.

Table 5.28: State variables for ORTHOTROPIC RATE Model (Section 5.2.31)

Index	Name	Variable Description
1	CRUSH	minimum volume ratio, crush is unrecoverable ($\hat{\epsilon}_V$)

5.2.31.1 Warnings and Usage Guidelines



Warning: Strongly rate-dependent models may fare poorly in implicit quasistatic solution. In implicit analyses the rate term used to evaluate the current load step is the rate seen by the model in the previous load step. This may cause the solution to oscillate between high- and low-rate equilibrium states from step to step.

5.2.32 Plane Stress Rate Plasticity

```
BEGIN PARAMETERS FOR MODEL PLANE_STRESS_RATE_PLASTICITY
#
# Elastic constants
#
YOUNGS MODULUS = <real>  $E$ 
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real>  $G$ 
BULK MODULUS   = <real>  $K$ 
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
# Isotropic/Kinematic hardening factor
#
BETA = <real>beta_parameter(1.0)
TOLERANCE = <real>tolerance(1.0e-10)
#
USER RATE DEPENDENCE = 0|1(0)
YIELD STRESS = <real>yield_stress
#
# input options for USER RATE DEPENDENCE = 0
#
# linear hardening
HARDENING MODULUS = <real>hardening_modulus
# power law hardening
HARDENING CONSTANT = <real>hardening_constant
HARDENING EXPONENT = <real>hardening_exponent(0.5)
# multilinear hardening
HARDENING FUNCTION = <string>hardening_function_name
# Johnson-cook rate dependence
RATE CONSTANT = <real>rate_constant
# multilinear rate dependence
RATE FUNCTION = <string>rate_function_name
#
# input options for USER RATE DEPENDENCE = 1
#
# rate-dependent yield
YIELD STRAIN RATES = <real_list>yield_strain_rates
YIELD CURVES = <string list>yield_function_names
FORMULATION = <int>formulation(0)
# rate-dependent damage
FRACTURE STRAIN = <real>fail_strain
FRACTURE STRAIN RATES = <real_list>fracture_strain_rates
FRACTURE CURVES = <string list>fracture_function_names
DECAY COEFFICIENT = <real>decay_coefficient(1.0)
#
# Thermal softening commands
```



```

INITIAL TEMPERATURE = <real>init_temp
MELT TEMPERATURE = <real>melt_temp
REFERENCE TEMPERATURE = <real>ref_temp
THERMAL EXPONENT = <real>therm_exp
#
# Unused commands
STRENGTH DIFFERENTIAL = <int>strength_diff
STRENGTH DIFFERENTIAL CURVES = <string>diff_curves
END [PARAMETERS FOR MODEL PLANE_STRESS_RATE_PLASTICITY]

```

The plane stress rate plasticity model is used only by shell elements.

See Section 5.1.5 for more information on elastic constants input.

The beta parameter defines if hardening is isotropic or kinematic. See Section 5.1.6 for details on the beta parameter.

When in the plastic regime an iterative scheme is applied in the return mapping algorithm to determine the radius of the yield surface. The convergence tolerance for this scheme is set by the TOLERANCE command line. The default TOLERANCE is 1.0e-10.

The initial yield stress must be defined by the YIELD STRESS command line.

The plane stress rate plasticity model has two separate sets of input commands defined by the USER RATE DEPENDENCE parameter. If USER RATE DEPENDENCE is set to zero the hardening model is defined by linear hardening, rate dependent power law hardening, or rate dependent multilinear hardening. Also, when USER RATE DEPENDENCE is set to zero, damage is not modeled. Alternatively, if USER RATE DEPENDENCE is set to one, the hardening model is defined by a series of yield curves respective to a series of strain rates, and the damage model is defined by a series of fracture strain curves respective to another series of strain rates. By default, USER RATE DEPENDENCE is set to zero.

5.2.32.1 Default Behavior: USER RATE DEPENDENCE = 0

A linear hardening model will be applied if the HARDENING MODULUS command line is given and the value provided for the hardening modulus is non-negative.

A power law hardening model will be applied if: the hardening modulus is not defined; the HARDENING CONSTANT command line is given; and the value provided for the hardening constant is non-negative. The value of the hardening exponent is defined by the HARDENING EXPONENT command line. The default HARDENING EXPONENT is 0.5.

If the above conditions do not hold for either the linear hardening model or the power law hardening model, then a multilinear hardening model will be applied. The HARDENING FUNCTION must be used for the multilinear hardening model to define a function for the hardening curve. The HARDENING FUNCTION command line references the name of a function defined in a FUNCTION command line in the SIERRA scope that describes the hardening behavior of the material as stress versus equivalent plastic strain. This curve is used to evaluate the slope (hardening modulus) at a given equivalent plastic strain. The x-values of the function should be values of equivalent plastic strain while the y-values of the function can be either the increment of stress over the yield stress

or the actual stress at the corresponding equivalent plastic strain. If an analytic function is used to define the hardening in the `FUNCTION` command, then `DIFFERENTIATE EXPRESSION` must be defined in the `FUNCTION` command and supply the derivative of the hardening function with respect to the equivalent plastic strain.

When `USER RATE DEPENDENCE` is set to zero, the strain rate function can be defined as the Johnson–Cook rate model or as multilinear. The Johnson–Cook strain rate function will be applied if the `RATE CONSTANT` command line is given and the value provided for the rate constant is non-negative. For more information about the Johnson–Cook material model and the effect of this rate constant on the material behavior, see [15].

If the `RATE CONSTANT` command is not given or the input value is negative, then the material model expects a multilinear rate function, given by the `RATE FUNCTION` command. This command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope. The x-values are treated as strain rates and the y-values are treated as scaling parameters. The scaling parameter is used as a multiplier to either increase or decrease the radius of the yield surface. A multiplier less than 1.0 decreases the radius and a multiplier greater than 1.0 increases the radius.

5.2.32.2 Alternate Behavior: `USER RATE DEPENDENCE = 1`

For the `USER RATE DEPENDENCE` option, the yield stress and fracture strain are each defined by a surface. The user defines each surface by a discrete set of curves.

The yield stress surface is defined by the `YIELD STRAIN RATES` and `YIELD CURVES` command lines. The `YIELD STRAIN RATES` command line defines one or more reference effective strain rate values, $\dot{\bar{\epsilon}}_1 \cdots \dot{\bar{\epsilon}}_i \cdots \dot{\bar{\epsilon}}_n$, where effective strain rate is defined by

$$\dot{\bar{\epsilon}} = \sqrt{\frac{4}{3}(\dot{\epsilon}_{xx}^2 + \dot{\epsilon}_{yy}^2 + \dot{\epsilon}_{xx}\dot{\epsilon}_{yy} + \dot{\epsilon}_{xy}^2)}. \quad (5.151)$$

The `YIELD CURVES` command line defines an equal number of function names, where the i^{th} function name references a function that defines the reference yield curve for $\dot{\bar{\epsilon}} = \dot{\bar{\epsilon}}_i$. Each function name is the name of a function defined by a `FUNCTION` command block in the SIERRA scope. Each function defines a reference hardening curve for the yield stress, $\bar{\sigma}_{\dot{\bar{\epsilon}}_i}$, with equivalent plastic strain as the independent variable, i.e.,

$$\bar{\sigma}_{\dot{\bar{\epsilon}}_i} = f(\bar{\epsilon}_p), \quad (5.152)$$

where $\bar{\sigma}$ is the von Mises yield stress. At every time step of an analysis, both $\bar{\epsilon}_p$ and $\dot{\bar{\epsilon}}$ are computed. $\dot{\bar{\epsilon}}$ is then compared to each reference strain rate. If $\dot{\bar{\epsilon}}$ is between two reference strain rates, $\dot{\bar{\epsilon}}_i$ and $\dot{\bar{\epsilon}}_k$, then $\bar{\sigma}_{\dot{\bar{\epsilon}}_i}$ and $\bar{\sigma}_{\dot{\bar{\epsilon}}_k}$ are evaluated for $\bar{\epsilon}_p$. These two reference yield values are then interpolated to obtain the current yield stress. If $\dot{\bar{\epsilon}}$ is larger than the maximum reference strain rate, $\dot{\bar{\epsilon}}_{i,max}$, then $\bar{\sigma}_{\dot{\bar{\epsilon}}_{i,max}}$ is evaluated to determine the current yield stress; no extrapolation is performed. Similarly, if $\dot{\bar{\epsilon}}$ is less than the minimum reference strain rate, $\dot{\bar{\epsilon}}_{i,min}$, then $\bar{\sigma}_{\dot{\bar{\epsilon}}_{i,min}}$ is evaluated to determine the current yield stress.

If the `FORMULATION` command is set to 0 (the default) then the total strain rate is used. If `FORMULATION` is anything other than 0, then the equivalent plastic strain rate is used.

The onset of material softening is determined by evaluating a damage parameter,

$$d = \sum_{j=0}^n \frac{\bar{\epsilon}_p^j - \bar{\epsilon}_p^{j-1}}{\epsilon_f^j(T, \dot{\epsilon})}, \quad (5.153)$$

where $\bar{\epsilon}_p^j$ is the equivalent plastic strain evaluated at the j^{th} time step, T is the stress triaxiality, $\dot{\epsilon}$ is the effective strain rate, and ϵ_f^j is the fracture strain evaluated at the j^{th} time step. At $d = 1$, the fracture strain is set to a constant, $\epsilon_{f,cr} = \epsilon_f^{j(d=1)}$, and subsequent damage accumulation is defined by

$$d = \sum_{j=0}^{j(d=1)} \frac{\bar{\epsilon}_p^j - \bar{\epsilon}_p^{j-1}}{\epsilon_f^j(T, \dot{\epsilon})} + \sum_{j=j(d=1)}^n \frac{\bar{\epsilon}_p^j - \bar{\epsilon}_p^{j-1}}{\epsilon_{f,cr}}. \quad (5.154)$$

Also, when $d > 1$, the radius of the yield surface is decayed exponentially,

$$\bar{\sigma} = \bar{\sigma} e^{C(1-d)}, \quad (5.155)$$

where C is a material constant defined by the `DECAY COEFFICIENT` command.

The triaxiality and rate dependence of the fracture strain before $d = 1$ is described by a surface, discretely defined by the `FRACTURE STRAIN RATES` and `FRACTURE CURVES` command lines. The `FRACTURE STRAIN RATES` command line defines one or more reference effective strain rate values, $\dot{\epsilon}_1 \cdots \dot{\epsilon}_i \cdots \dot{\epsilon}_n$. The `FRACTURE CURVES` command line defines an equal number of function names, where the i^{th} function name references a function that defines the reference fracture strain curves for $\dot{\epsilon} = \dot{\epsilon}_i$. Each function name is the name of a function defined by a `FUNCTION` command block in the SIERRA scope. Each function defines a reference curve for the fracture strain, $\epsilon_{f,\dot{\epsilon}_i}$, with stress triaxiality as the independent variable, i.e.,

$$\epsilon_{f,\dot{\epsilon}_i} = f(T). \quad (5.156)$$

The `FRACTURE STRAIN` command defines the nominal equivalent plastic strain at which fracture occurs.

At every time step of an analysis, both T and $\dot{\epsilon}$ are computed. $\dot{\epsilon}$ is then compared to each reference strain rate. If $\dot{\epsilon}$ is between two reference strain rates, $\dot{\epsilon}_i$ and $\dot{\epsilon}_k$, then $\epsilon_{f,\dot{\epsilon}_i}$ and $\epsilon_{f,\dot{\epsilon}_k}$ are evaluated for T . These two reference fracture strain values are then interpolated to get the current fracture strain. If $\dot{\epsilon}$ is larger than the maximum reference strain rate, $\dot{\epsilon}_{i,max}$, then $\bar{\sigma}_{\dot{\epsilon}_{i,max}}$ is evaluated to determine the current fracture strain; no extrapolation is performed. Similarly, if $\dot{\epsilon}$ is less than the minimum reference strain rate, $\dot{\epsilon}_{i,min}$, then $\bar{\sigma}_{\dot{\epsilon}_{i,min}}$ is evaluated to determine the current fracture strain.

Material-dependent element death can also be activated for `USER RATE DEPENDENCE = 1` by using the material death criterion described in Section 6.5. The element is killed when the value of $C(1 - d)$ from Equation (5.155) gets below a critical value. This critical value is currently set to -10.0 .

5.2.32.3 Other Options

An additional thermal softening capability is available in this model. Given a material state temperature T , melt temperature T_{melt} , reference temperature T_{ref} , and thermal exponent n the additional

thermal softening factor α is defined by Equation 5.157.

$$\alpha = 1.0 - \left(\frac{T - T_{ref}}{T_{melt} - T_{ref}} \right)^n \quad (5.157)$$

The INITIAL TEMPERATURE defines the initial material temperature. Note the thermal softening of this material is NOT linked to the standard model “temperature” field.

MELT TEMPERATURE defines T_{melt} in Equation 5.157.

REFERENCE TEMPERATURE defines T_{ref} in Equation 5.157.

THERMAL EXPONENT defines n in Equation 5.157.

FORMULATION controls the strain rate source term. A value of 0 triggers the use of the total strain rate; any other value uses the plastic strain rate.

The state variables for this model are listed in Table 5.29

Table 5.29: State Variables for ELASTIC PLASTIC Model (Section 5.2.32)

Index	Name	Variable Description
1	EQPS	equivalent plastic strain
2	RADIUS	radius of yield surface
9	TEMPERATURE	material internal temperature field

5.2.32.4 Warnings and Usage Guidelines



Warning: Strongly rate-dependent models may fare poorly in implicit quasistatic solution. In implicit analyses the rate term used to evaluate the current load step is the rate seen by the model in the previous load step. This may cause the solution to oscillate between high- and low-rate equilibrium states from step to step.

5.2.33 Incompressible Solid Model

```
BEGIN PARAMETERS FOR MODEL INCOMPRESSIBLE_SOLID
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real>  $\lambda$ 
TWO MU         = <real>  $2\mu$ 
#
#
#
K SCALING = <real>k_scaling
2G SCALING = <real>2g_scaling
TARGET E = <real>target_e
MAX POISSONS RATIO = <real>max_poissons_ratio
REFERENCE STRAIN = <real>reference_strain
SCALING FUNCTION = <string>scaling_function_name
END [PARAMETERS FOR MODEL INCOMPRESSIBLE_SOLID]
```

The incompressible solid model is a variation of the elastic model and can be used in both Presto and Adagio. In Adagio, the incompressible solid model is used with the control-stiffness option in the multilevel solver. The control-stiffness option is implemented via the `CONTROL STIFFNESS` command block and is discussed in Chapter 4. The model is used to model nearly incompressible materials where Poisson’s ratio, ν , ≈ 0.5 . In the course of solving a series of model problems in Adagio, the material response from this model incorporates scaling the bulk and/or shear behaviors to yield a material response that is more amenable to solution using Adagio’s conjugate gradient solver. The final material behavior that is calculated corresponds to the actual moduli that are specified. When this model is used in Presto, the material scalings are ignored, and the model behaves like a linear elastic model.

In the above command blocks, the following definitions are applicable. Usage requirements are identified both in this list of definitions and in the discussion that follows the list.

- See Section 5.1.5 for more information on elastic constants input.
- The following material-scaling command lines are used only in Adagio and are optional as described below:
 - The nominal bulk scaling is defined with the `K SCALING` command line.
 - The nominal shear scaling is defined with the `2G SCALING` command line.
 - The target Young’s modulus is defined with the `TARGET E` command line.
 - The maximum Poisson’s ratio is defined with the `MAX POISSONS RATIO` command line.
 - The reference strain is defined with the `REFERENCE STRAIN` command line.
 - The `SCALING FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the time dependent

scaling to be applied.

As noted previously, only two of the elastic constants are required to define the unscaled material response. This requirement applies to use of the incompressible solid model in Presto and in Adagio. Further, all the material-scaling command lines are only used in Adagio.

Several options exist for defining the bulk and/or shear scalings that can be used with the multilevel solver in Adagio.

- Option 1: You can provide the scalings directly by including both of the `K SCALING` and `2G SCALING` command lines or either of them. When both command lines are input, the user-specified values for their parameters will be used. If only the `K SCALING` command line is input, the bulk scaling is as specified in the `k_scaling` parameter, and the value of the shear scaling parameter, `2g_scaling`, is set to 1.0. On the other hand, if only the `2G SCALING` command line is input, then the shear scaling is as specified in the `2g_scaling` parameter, but the value of the bulk-scaling parameter, `k_scaling`, is not set to 1.0. Instead, the bulk scaling is determined by computing a scaled bulk modulus from the scaled shear modulus and a (scaled) Poisson's ratio of 0.3. Then, the bulk scaling is determined simply as the ratio of the scaled bulk modulus to the actual bulk modulus.
- Option 2: You can specify either or both of the `TARGET E` and `MAX POISSONS RATIO` command lines to define the scalings. If only the `TARGET E` command line is included, the bulk and shear scalings are computed by first finding scaled moduli using the value of the `target_e` parameter along with a (scaled) Poisson's ratio of 0.3. The bulk and shear scalings are then determined as the ratio of the appropriate scaled to unscaled modulus. If only the `MAX POISSONS RATIO` command line is included, the shear scaling is set to 1.0, and the bulk scaling is computed by first calculating a scaled bulk modulus from the actual shear modulus and the value of the `max_poissons_ratio` parameter. The bulk scaling is then calculated simply as the ratio of the scaled bulk modulus to the actual bulk modulus. If both the `TARGET E` and `MAX POISSONS RATIO` command lines are included, the bulk scaling (and shear scaling) is determined from the ratio of the bulk scaled modulus (and shear scaled modulus) computed using the values of the `target_e` and `max_poissons_ratio` parameters to the unscaled bulk (and shear) modulus.
- Option 3: You can choose not to include any of the `K SCALING`, `2G SCALING`, `TARGET E`, and `MAX POISSONS RATIO` command lines. In such case, the shear scaling is set to 1.0, and the bulk scaling is computed as the ratio of the scaled bulk modulus coming from the real shear modulus and a (scaled) Poisson's ratio of 0.3 to the actual bulk modulus.

The function referenced by the value of the parameter `scaling_function_name` in the `SCALING FUNCTION` command line can be used to modify the bulk and shear scalings in solution time. The actual scalings used are computed by taking the scalings specified by the parameter values in the `K SCALING`, `2G SCALING`, `TARGET E`, and `MAX POISSONS RATIO` command lines and simply multiplying them by the function value at the specified solution time. If the `SCALING FUNCTION` command line is not included, the bulk and shear scalings are fixed in time.

The `REFERENCE STRAIN` command line supplies a value for the reference strain that is used to create a normalized material constraint violation based on strains. Specifying a reference strain implies the use of strains for measuring the material constraint violation (or part of the control-stiffness error in Adagio). Otherwise, the material constraint violation is determined by using the

change in the scaled stress response over the current model problem.

5.2.34 Mooney-Rivlin Model

```
BEGIN PARAMETERS FOR MODEL MOONEY_RIVLIN
#
# Elastic constants, One of these elastic constants must be input.
# The shear modulus is computed from C10 and C01.
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
#
#
C10 = <real>c10
C01 = <real>c01
C10 FUNCTION = <string>c10_function_name
C01 FUNCTION = <string>c01_function_name
BULK FUNCTION = <string>bulk_function_name
THERMAL EXPANSION FUNCTION = <string>eth_function_name
TARGET E = <real>target_e
TARGET E FUNCTION = <string>etar_function_name
MAX POISSONS RATIO = <real>max_poissons_ratio(0.5)
REFERENCE STRAIN = <real>reference_strain
END [PARAMETERS FOR MODEL MOONEY_RIVLIN]
```

Mooney-Rivlin is a hyperelastic material model that is used to describe rubber. The Mooney-Rivlin model incorporates temperature-dependent material moduli and can be used in both Presto and Adagio. When the model is used in Adagio, it can be used with or without the control-stiffness option in Adagio's multilevel solver. The control-stiffness option is implemented via the `CONTROL STIFFNESS` command block and is discussed in Chapter 4. The model is used to simulate nearly incompressible materials where Poisson's ratio, ν , ≈ 0.5 . In the course of solving a series of model problems in Adagio, the material response from this model incorporates scaling the bulk and/or shear behaviors to yield a material response that is more amenable to solution using Adagio's conjugate gradient solver. The final material behavior that is calculated corresponds to the actual moduli that are specified. When this model is used in Presto, the material scalings are ignored.

In the above command blocks the following command lines are required:

- The material constants C10 and C01 determine the shear behavior as defined by

$$shear\_modulus = 2(C_{10} + C_{01}). \quad (5.158)$$

Thus the C10 and C01 command lines must be included in this model and the shear modulus elastic constant is not an available input.

- Since C10 and C01 define the shear behavior, one additional elastic constant, other than the shear modulus is required to define the unscaled bulk behavior. See Section 5.1.5 for more information on elastic constants input.

Other command line options include:

- The `C10 FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the temperature dependence of the C10 material parameter. This command line is optional. If it is not present, there is no temperature dependence in the C10 parameter. See the usage discussion below.
- The `C01 FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the temperature dependence of the C01 material parameter. This command line is optional. If it is not present, there is no temperature dependence in the C01 parameter. See the usage discussion below.
- The `BULK FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the temperature dependence of the bulk modulus. This command line is optional. If it is not present, there is no temperature dependence in the bulk modulus. See the usage discussion below.
- The `THERMAL EXPANSION FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the linear thermal expansion as function of temperature. This command line is optional. If it is not present, there is no thermal expansion. See the usage discussion below.
- The following material-scaling command lines are used only in Adagio:
 - The target Young's modulus is defined with the `TARGET E` command line. This command line is optional. See the usage discussion below.
 - The `TARGET E FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the time variation of the target Young's modulus. This command line is optional. If it is not present, there is no time dependence in the Target E parameter. See the usage discussion below.
 - The maximum Poisson's ratio is defined with the `MAX POISSONS RATIO` command line. This command line is optional and will default to 0.5 if not specified. See the usage discussion below.
 - The reference strain is defined with the `REFERENCE STRAIN` command line. This command line is optional. See the usage discussion below.

The command lines for functions that specify the temperature dependence of C10, C01, and bulk modulus are optional, e.g., the `C10 FUNCTION`, `C01 FUNCTION` and `BULK FUNCTION` command lines. If these command lines are not included, their corresponding material parameters are taken to be independent of temperature. Mooney-Rivlin, like other material models, allows for the specification of thermal strain behavior within the material model itself, via the `THERMAL EXPANSION FUNCTION` command line. This command line, like the other “function-type” command lines in this model requires that a function associated with the name be defined in the SIERRA scope.

The bulk and shear scalings that can be used with the multilevel solver in Adagio are specified via a combination of the `TARGET E`, `TARGET E FUNCTION`, and `MAX POISSONS RATIO` command lines. If the `TARGET E` command line is not included (and the `MAX POISSONS RATIO` command line is included), the shear scaling is set to 1.0, and the bulk scaling is determined from the ratio of the scaled bulk modulus to its unscaled value, where the scaled bulk modulus is computed using the value of the `max_poissons_ratio` parameter along with the unscaled initial shear modulus that

is determined from the value of the parameters specified in the C10 and C01 command lines. On the other hand, if both the TARGET E command line and the MAX POISSONS RATIO command line are included, bulk and shear scaling values are computed using scaled moduli that are calculated from the target_e and max_poissons_ratio parameter values.

Including the TARGET E FUNCTION command line allows time-dependent bulk and shear scaling to be used. If this command line is not specified, the bulk and shear scalings remain constant in solution time. If the command line is specified, the target Young's modulus that is used for computing the scaled moduli is multiplied by the function value.

The REFERENCE STRAIN command line supplies a value for the reference strain used to create a normalized material constraint violation that is based on strains. Specifying a reference strain implies the use of strains for measuring the material constraint violation (or part of the control-stiffness error in Adagio). Otherwise, the material constraint violation is determined using the change in the scaled stress response over the current model problem.

Brief documentation on the theoretical basis for the Mooney-Rivlin model is given in [1].

State variables for this model are listed in Table 5.30.

Table 5.30: State Variables for MOONEY RIVLIN Model (Section 5.2.34)

Name	Name	Variable Description
1	C10	material constant
2	C01	material constant
3	K	temperature scaled bulk modulus
4	SFJTH	total volumetric strain
5	JTH	thermal volumetric strain
6	SFJTH_FLAG	thermal expansion flag (1 if thermally expanding)
7	WDEV	distortion strain energy
8	SHEAR MODULUS	temperature scaled shear modulus calculated from C10 and C01

5.2.35 Stiff Elastic

```
BEGIN PARAMETERS FOR MODEL STIFF_ELASTIC
#
# Elastic constants
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real>  $\nu$ 
SHEAR MODULUS  = <real> G
BULK MODULUS   = <real> K
LAMBDA         = <real>  $\lambda$ 
TWO MU        = <real>  $2\mu$ 
#
#
#
SCALE FACTOR = <real>scale_factor
REFERENCE STRAIN = <real>reference_strain
END [PARAMETERS FOR MODEL STIFF_ELASTIC]
```

The stiff elastic model is a variation of the isotropic elastic model. The stiff elastic model can be used in both implicit and explicit analysis. However, model is primarily intended to be used with implicit solution in conjunction with the control-stiffness multilevel solver. The control-stiffness option is implemented via the `CONTROL STIFFNESS` command block and is discussed in Chapter 4. The stiff elastic model is used slowly ramp of the stiffness of a stiff material during control stiffness solution to generate a material response more amenable to solution using the implicit conjugate gradient solver. The ultimate material stiffness that is used in the final control stiffness iteration corresponds to the actual moduli that are specified. Essentially when used with control stiffness this model provides an series of predictions and corrections of increasingly stiff materials. It is easier to solve this series of prediction analyses than to directly solve the original poorly conditioned full stiffness analysis.

When this model is used in explicit analysis, or without the control stiffness solver, the material scalings are ignored.

In the above command blocks:

- See Section 5.1.5 for more information on elastic constants input. These constants define the true material stiffness.
- The scaled bulk and shear moduli are computed using a Young's modulus scaled by the value given by the `SCALE FACTOR` line command. These are the scaled down moduli used at the start of the control stiffness iteration.
- The `REFERENCE STRAIN` command line supplies a value for the reference strain used to create a normalized material constraint violation that is based on strains. Specifying a reference strain implies the use of strains for measuring the material constraint violation (or part of the control-stiffness error in Adagio). Otherwise, the material constraint violation is determined using the change in the scaled stress response over the current model problem. Specifying a smaller reference strain means the material constraint is enforced more accurately.

This material has no state variables.

5.2.36 Swanson Model

```
BEGIN PARAMETERS FOR MODEL SWANSON
#
# Elastic constants, the shear modulus is computed
# from A1, P1, B1, Q1, C1, and R1.
# One of these elastic constants must be input.
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real>  $\nu$ 
BULK MODULUS   = <real> K
LAMBDA         = <real>  $\lambda$ 
#
#
A1 = <real>a1
P1 = <real>p1
B1 = <real>b1
Q1 = <real>q1
C1 = <real>c1
R1 = <real>r1
CUT OFF STRAIN = <real>ecut
THERMAL EXPANSION FUNCTION = <string>eth_function_name
TARGET E = <real>target_e
TARGET E FUNCTION = <string>etar_function_name
MAX POISSONS RATIO = <real>max_poissons_ratio
REFERENCE STRAIN = <real>reference_strain
END [PARAMETERS FOR MODEL SWANSON]
```

The Swanson material model is a hyperelastic constitutive model that is used to simulate rubber. The Swanson model can be used in both Presto and Adagio. When the model is used in Adagio, it can be used with or without the control-stiffness option in Adagio's multilevel solver for nearly incompressible materials where Poisson's ratio, ν , ≈ 0.5 . The control-stiffness option is implemented via the `CONTROL STIFFNESS` command block and is discussed in Chapter 4. In the course of solving a series of model problems in Adagio, the material response from this model incorporates scaling the bulk and/or shear behaviors to yield a material response that is more amenable to solution using Adagio's conjugate gradient solver. The final material behavior that is calculated corresponds to the actual moduli that are specified. When this model is used in Presto, the material scalings are ignored.

In the above command blocks the following command lines are required:

- The `A1` command line defines A_1 in Equation 5.159.
- The `P1` command line defines P_1 in Equation 5.159.
- The `B1` command line defines B_1 in Equation 5.159.
- The `Q1` command line defines Q_1 in Equation 5.159.
- The `C1` command line defines C_1 in Equation 5.159.
- The `R1` command line defines R_1 in Equation 5.159.
- The small-strain value used for computing the initial shear modulus is defined with the `CUT`

OFF STRAIN command line. This defines e_c in Equation 5.160.

- The above properties define the shear behavior. One additional elastic constant other than the shear modulus is required to define the unscaled bulk behavior. See Section 5.1.5 for more information on elastic constants input.

Other command lines include:

- The THERMAL EXPANSION FUNCTION command line references the name of a function defined in a FUNCTION command line in the SIERRA scope that describes the linear thermal expansion as a function of temperature. This command line is optional. If it is not present, there is no thermal expansion. See the usage discussion below.
- The following material-scaling command lines are used only in Adagio:
 - The target Young's modulus is defined with the TARGET E command line. This command line is optional. See the usage discussion below.
 - The TARGET E FUNCTION command line references the name of a function defined in a FUNCTION command line in the SIERRA scope that describes the time variation of the target Young's modulus. This command line is optional. If it is not present, there is no time dependence in the Target E parameter. See the usage discussion below.
 - The maximum Poisson's ratio is defined with the MAX POISSONS RATIO command line. This command line is optional and will default to 0.5 if not specified. See the usage discussion below.
 - The reference strain is defined with the REFERENCE STRAIN command line. This command line is optional. See the usage discussion below.

The strain energy density for the Swanson model is given by

$$U = \frac{3}{2} \frac{A_1}{P_1 + 1} \left(\frac{\bar{I}_1}{3} - 1 \right)^{P_1+1} + \frac{3}{2} \frac{B_1}{Q_1 + 1} \left(\frac{\bar{I}_2}{3} - 1 \right)^{Q_1+1} + \frac{3}{2} \frac{C_1}{R_1 + 1} \left(\frac{\bar{I}_1}{3} - 1 \right)^{R_1+1} + K (J_m \ln (J_m) - J_m,) \quad (5.159)$$

where $\bar{I}_1$ and $\bar{I}_2$ are the first and second invariants of the left Cauchy-Green strain tensor $\bar{\mathbf{B}}$ which excludes volume change and thermal strains, J_m is the mechanical only part of the volume change which excludes thermal strains, and K is the bulk modulus.

As noted previously, only one of the standard elastic constants are required to define the unscaled bulk behavior. Together, the values for parameters in the A1, P1, B1, Q1, C1, and R1 command lines define the unscaled shear behavior, so these command lines must be present. The initial unscaled shear modulus is determined from those parameter values along with the value of the parameter in the CUT OFF STRAIN command line via

$$shear\_modulus = A_1 \left(\frac{e_c^2}{3} \right)^{P_1} + 2B_1 \left(\frac{e_c^2}{3} \right)^{Q_1} \quad (5.160)$$

The Swanson model, like a few of the material models, allows for the specification of thermal strain behavior within the material model itself, via the THERMAL EXPANSION FUNCTION command line. This command line, like the other “function-type” command lines in this model, requires that a function associated with the name be defined in the SIERRA scope.

The bulk and shear scalings that can be used with the multilevel solver in Adagio are specified via a combination of the `TARGET E`, `TARGET E FUNCTION`, and `MAX POISSONS RATIO` command lines. If the `TARGET E` command line is not included (and the `MAX POISSONS RATIO` command line is included), the shear scaling is set to 1.0, and the bulk scaling is determined from the ratio of the scaled bulk modulus to its unscaled value, where the scaled bulk modulus is computed using the value of the `max_poissons_ratio` parameter along with the unscaled shear modulus. On the other hand, if both the `TARGET E` command line and the `MAX POISSONS RATIO` are included, bulk and shear scaling values are computed using scaled moduli that are calculated from the `target_e` and `max_poissons_ratio` parameter values.

Including the `TARGET E FUNCTION` command line allows time-dependent bulk and shear scaling to be used. If this command line is not specified, the bulk and shear scalings remain constant in solution time. If the command line is specified, the target Young's modulus that is used for computing the scaled moduli is multiplied by the function value.

The `REFERENCE STRAIN` command line supplies a value for the reference strain used to create a normalized material constraint violation that is based on strains. Specifying a reference strain implies the use of strains for measuring the material constraint violation (or part of the control-stiffness error in Adagio). Otherwise, the material constraint violation is determined using the change in the scaled stress response over the current model problem.

Output variables available for this model are listed in Table 5.31. Brief documentation on the theoretical basis for the Swanson model is given in [1].

Table 5.31: State Variables for SWANSON Model (Section 5.2.36)

Index	Name	Variable Description
1	SFJTH	total volumetric strain
2	JTH	thermal volumetric strain
3	VMECHXX	mechanical left stretch xx component
4	VMECHYY	mechanical left stretch yy component
5	VMECHZZ	mechanical left stretch zz component
6	VMECHXY	mechanical left stretch xy component
7	VMECHYZ	mechanical left stretch yz component
8	VMECHZX	mechanical left stretch zx component
9	SFJTH_FLAG	thermal expansion flag (1 if thermally expanding)
10	WDEV	distortion strain energy

5.2.37 Viscoelastic Swanson Model

```
BEGIN PARAMETERS FOR MODEL VISCOELASTIC_SWANSON
#
# Elastic constants, note shear modulus is computed
# from A1, P1, B1, Q1, C1, and R1.
# One of these elastic constants must be input.
#
YOUNGS MODULUS = <real> E
POISSONS RATIO = <real> ν
BULK MODULUS   = <real> K
LAMBDA         = <real> λ
#
#
BULK FUNCTION <string>bulk_func
A1 = <real>a1
A1 FUNCTION <string>a1_func
P1 = <real>p1
P1 FUNCTION <string>p1_func
B1 = <real>b1
B1 FUNCTION <string>b1_func
Q1 = <real>q1
Q1 FUNCTION <string>q1_func
C1 = <real>c1
C1 FUNCTION <string>c1_func
R1 = <real>r1
R1 FUNCTION <string>r1_func
CUT OFF STRAIN = <real>ecut
THERMAL EXPANSION FUNCTION = <string>eth_function_name
PRONY SHEAR INFINITY = <real>ginf
PRONY SHEAR 1 = <real>g1
PRONY SHEAR 2 = <real>g2
PRONY SHEAR 3 = <real>g3
PRONY SHEAR 4 = <real>g4
PRONY SHEAR 5 = <real>g5
PRONY SHEAR 6 = <real>g6
PRONY SHEAR 7 = <real>g7
PRONY SHEAR 8 = <real>g8
PRONY SHEAR 9 = <real>g9
PRONY SHEAR 10 = <real>g10
SHEAR RELAX TIME 1 = <real>tau1
SHEAR RELAX TIME 2 = <real>tau2
SHEAR RELAX TIME 3 = <real>tau3
SHEAR RELAX TIME 4 = <real>tau4
SHEAR RELAX TIME 5 = <real>tau5
SHEAR RELAX TIME 6 = <real>tau6
SHEAR RELAX TIME 7 = <real>tau7
SHEAR RELAX TIME 8 = <real>tau8
```



```

SHEAR RELAX TIME 9 = <real>tau9
SHEAR RELAX TIME 10 = <real>tau10
WLF COEF C1 = <real>wlf_c1
WLF COEF C2 = <real>wlf_c2
WLF TREF = <real>wlf_tref
NUMERICAL SHIFT FUNCTION = <string>ns_function_name
TARGET E = <real>target_e
TARGET E FUNCTION = <string>etar_function_name
MAX POISSONS RATIO = <real>max_poissons_ratio(0.5)
REFERENCE STRAIN = <real>reference_strain
SHEAR MODULUS UPDATE PENALTY = <real>Sherwood
END [PARAMETERS FOR MODEL VISCOELASTIC_SWANSON]

```

The viscoelastic Swanson model is a finite strain viscoelastic model that has an initial elastic response that matches the Swanson material model. The bulk response is elastic, while the shear response is viscoelastic. This model is commonly employed in simulating the response of rubber materials. The viscoelastic Swanson model can be used in both Presto and Adagio. When the model is used in Adagio, it can be used with or without the control-stiffness option in Adagio's multilevel solver for nearly incompressible materials where Poisson's ratio, ν , ≈ 0.5 . The control-stiffness option is implemented via the `CONTROL STIFFNESS` command block and is discussed in Chapter 4. By solving a series of model problems in Adagio, the material response from this model incorporates scaling the bulk and/or shear behaviors to yield a material response that is more amenable to solution using Adagio's conjugate gradient solver. The final material behavior that is calculated corresponds to the actual moduli that are specified. When this model is used in Presto, the material scalings are ignored.

In the above command blocks the following command lines are required:

- Material constants command lines `A1`, `P1`, `B1`, `Q1`, `C1`, `R1`.
- The small-strain value used for computing the glassy shear modulus is defined with the `CUT OFF STRAIN` command line. This defines e_c in Equation 5.161.
- The above parameters define the unscaled shear modules as defined by Equation 5.161. One other elastic constant needs to be included to define the unscaled bulk behavior. The other elastic constant cannot be the shear modulus. See Section 5.1.5 for more information on elastic constants input.
- `PRONY SHEAR INFINITY` command line.
- `WLF COEF C1` command line.
- `WLF COEF C2` command line.
- `WLF TREF` command line.

Other optional command lines are:

$$shear\_modulus = A_1 \left(\frac{e_c^2}{3} \right)^{P_1} + 2B_1 \left(\frac{e_c^2}{3} \right)^{Q_1} \quad (5.161)$$

- The temperature variation of material constants `A1`, `P1`, `B1`, `Q1`, `C1`, `R1` may be defined with their respective functions `A1 FUNCTION`, `P1 FUNCTION`, `B1 FUNCTION`, `Q1 FUNCTION`, `C1 FUNCTION`, and `R1 FUNCTION`.

- The temperature variation of the `BULK MODULUS` constant may be defined with the `BULK FUNCTION` command line.
- The `THERMAL EXPANSION FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the linear thermal expansion as a function of temperature. This command line is optional. If it is not present, there is no thermal expansion. See the usage discussion below.
- The normalized relaxation spectra coefficients are specified with the `PRONY SHEAR I` command lines, where the value of `I` varies sequentially from 1 to 10.
- The normalized relaxation spectra time constants are specified with the `SHEAR RELAX TIME I` command lines, where the value of `I` varies sequentially from 1 to 10.
- `NUMERICAL SHIFT FUNCTION` command line.
- The following material-scaling command lines are used only in Adagio:
 - The target Young's modulus is defined with the `TARGET E` command line. This command line is required. See the usage discussion below.
 - The `TARGET E FUNCTION` command line references the name of a function defined in a `FUNCTION` command line in the SIERRA scope that describes the time variation of the target Young's modulus. This command line is optional. If it is not present, there is no time dependence in the Target E parameter. See the usage discussion below.
 - The maximum Poisson's ratio is defined with the `MAX POISSONS RATIO` command line. This command line is optional and will default to 0.5 if not specified. See the usage discussion below.
 - The reference strain is defined with the `REFERENCE STRAIN` command line. This command line is required. See the usage discussion below.

As noted previously, only two of the elastic constants are required to define the unscaled bulk behavior. Together, the values for parameters in the `A1`, `P1`, `B1`, `Q1`, `C1`, and `R1` command lines define the unscaled glassy shear behavior, so these command lines must be present. The unscaled glassy shear modulus is determined from those parameter values along with the value of the parameter in the `CUT OFF STRAIN` command line, so this command line is also required.

The viscoelastic Swanson model, like a few of the material models, allows for the specification of thermal strain behavior within the material model itself, via the `THERMAL EXPANSION FUNCTION` command line. This command line, like the other “function-type” command lines in this model requires that a function associated with the name be defined in the SIERRA scope.

The bulk and shear scalings that can be used with the multilevel solver in Adagio are specified via a combination of the `TARGET E`, `TARGET E FUNCTION`, and `MAX POISSONS RATIO` command lines. If the `TARGET E` command line is not included (and the `MAX POISSONS RATIO` command line is included), the shear scaling is set to 1.0, and the bulk scaling is determined from the ratio of the scaled bulk modulus to its unscaled value, where the scaled bulk modulus is computed using the value of the `max_poissons_ratio` parameter along with the unscaled shear modulus. On the other hand, if both the `TARGET E` command line and the `MAX POISSONS RATIO` command line are included, bulk and shear scaling values are computed using scaled moduli that are calculated from the `target_e` and `max_poissons_ratio` parameter values.

Including the `TARGET E FUNCTION` command line allows time-dependent bulk and shear scaling to be used. If this command line is not specified, the bulk and shear scalings remain constant

in solution time. If the command line is specified, the target Young's modulus that is used for computing the scaled moduli is multiplied by the function value.

The `REFERENCE STRAIN` command line supplies a value for the reference strain used to create a normalized material constraint violation that is based on strains. Specifying a reference strain implies the use of strains for measuring the material constraint violation (or part of the control-stiffness error in Adagio). Otherwise, the material constraint violation is determined using the change in the scaled stress response over the current model problem.

The `SHEAR MODULUS UPDATE PENALTY` Is associated with how the viscoelastic Swanson model interacts with control stiffness. This command is not recommended for use at this time.

Output variables available for this model are listed in Table [5.32](#). Brief documentation on the theoretical basis for the viscoelastic Swanson model is given in [[1](#), [41](#), [42](#), [43](#)].

Table 5.32: State Variables for VISCOELASTIC SWANSON Model (Section 5.2.37)

Index	Name	Variable Description
1	SFJTH	total volumetric strain
2	JTH	thermal volumetric strain
3	VMECHXX	mechanical left stretch xx
4	VMECHYY	mechanical left stretch yy
5	VMECHZZ	mechanical left stretch zz
6	VMECHXY	mechanical left stretch xy
7	VMECHYZ	mechanical left stretch yz
8	VMECHZX	mechanical left stretch zx
9	VSXXDEV1 – VSXXDEV10	viscoelastic deviatoric PK2 stress xx
10	VSYYDEV1 – VSYYDEV10	viscoelastic deviatoric PK2 stress yy
11	VSZZDEV1 – VSZZDEV10	viscoelastic deviatoric PK2 stress zz
12	VSXYDEV1 – VSXYDEV10	viscoelastic deviatoric PK2 stress xy
13	VSYZDEV1 – VSYZDEV10	viscoelastic deviatoric PK2 stress yz
14	VSZXDEV1 – VSZXDEV10	viscoelastic deviatoric PK2 stress zx
15	SOXXDEV	instantaneous deviatoric PK2 stress xx
16	SOYYDEV	instantaneous deviatoric PK2 stress yy
17	SOZZDEV	instantaneous deviatoric PK2 stress zz
18	SOXYDEV	instantaneous deviatoric PK2 stress xy
19	SOYZDEV	instantaneous deviatoric PK2 stress yz
20	SOZXDEV	instantaneous deviatoric PK2 stress zx
21	WLF_AAVG	average WLF shift factor
22	ANAVG	average shift factor
23	WDEV	distortion strain energy

5.2.38 Additional Material Models in Development

In addition to the material models documented here, many additional material models are documented in the Sierra/SolidMechanics 4.46 Capabilities in Development manual. These models are listed in Table 5.33.

The models, while available for use in Sierra/SM, can be used with the realization that they may not be fully verified or supported. We will periodically move models that are in development to production after they have been documented and tested. Once a model is in production it will be fully documented in this user manual.

Table 5.33: Material models documented in Sierra/SolidMechanics 4.46 Capabilities in Development

Material Models in Development	
ELASTIC ORTHOTROPIC DAMAGE	NLVE 3D ORTHOTROPIC
KARAGOZIAN AND CASE CONCRETE	HONEYCOMB
KAYENTA	FOAM DAMAGE
SHAPE MEMORY ALLOY	PIEZO
LINEAR ELASTIC	THERMO EP POWER
HOSFORD PLASTICITY	THERMO EP POWER WELD
KARAFILLIS BOYCE PLASTICITY	UNIVERSAL POLYMER
BARLAT PLASTICITY	

5.3 Cohesive Zone Material Models

Several material models are available for use with cohesive zone elements or with Dash contact. These models are described in this section.

Traction separation models used for cohesive surface elements are input within `BEGIN MATERIAL` blocks in the same manner as continuum models. Although density is not a property used by cohesive zone elements, because of their specification within this block, all of these models currently require a density to be provided as part of their input.

5.3.1 Traction Decay

```
BEGIN PARAMETERS FOR MODEL TRACTION_DECAY
  NORMAL DECAY LENGTH = <real>
  TANGENTIAL DECAY LENGTH = <real>
END [PARAMETERS FOR MODEL TRACTION_DECAY]
```

The Traction Decay cohesive model is a simple model that gets initialized with a traction upon activation or insertion of the cohesive element. The traction then decays to zero over specified values of normal and tangential separation. This model is only valid to use in conjunction with dynamic cohesive zone activation through MPC deactivation or dynamic insertion of cohesive surface elements.

In the above command block:

- The density of the material is defined with the `DENSITY` command line. Although this is not used in the model, it is currently a required input parameter.
- The separation length over which the normal traction decays to zero is set by the `NORMAL DECAY LENGTH` command line.
- The separation length over which the tangential traction decays to zero is set by the `TANGENTIAL DECAY LENGTH` command line.

The state variables for this model are listed in Table 5.3.1.

Table 5.34: State Variables for TRACTION DECAY Surface Model (Section 5.3.1)

Index	Name	Variable Description
0	MAX_SEPARATION_S	maximum separation in the first tangential direction
1	MAX_SEPARATION_T	maximum separation in the second tangential direction
2	MAX_SEPARATION_N	maximum separation in the normal direction

5.3.2 Tvergaard Hutchinson

```
BEGIN PARAMETERS FOR MODEL TVERGAARD_HUTCHINSON
  INIT TRACTION METHOD = IGNORE|ADD|EXTRINSIC (IGNORE)
  LAMBDA_1 = <real>
  LAMBDA_2 = <real>
  NORMAL LENGTH SCALE = <real>
  TANGENTIAL LENGTH SCALE = <real>
  PEAK TRACTION = <real>
  PENETRATION STIFFNESS MULTIPLIER = <real>
  USE ELASTIC UNLOADING = NO|YES (YES)
END [PARAMETERS FOR MODEL TVERGAARD_HUTCHINSON]
```

The Tvergaard Hutchinson cohesive model combines the normal and tangential separation into a single normalized separation and calculates a traction per unit length based on this value. This model then calculates normal and tangential traction based on the ratio of the normal and tangential length scales.

In the above command blocks:

- For dynamically activated or dynamically inserted cohesive surface elements, an initial traction can be used to initialize the traction-separation model based on element properties specified in the `ELEMENT DEATH` or `XFEM` block. This is set via the `INIT TRACTION METHOD`. The default behavior is to ignore any initial tractions and let the traction-separation law dictate the behavior. Note that this does not maintain equilibrium upon cohesive element insertion. The `ADD` method offsets the traction-separation law by the initiation traction, which is linearly scaled to zero under further loading. The `EXTRINSIC` method initializes the state of the traction-separation law along the hardening branch. In the case that the initialization traction is greater than the specified peak traction, the traction-separation law is evaluated from the beginning of the traction plateau with the peak traction value reset to the initialization traction. Both the `ADD` and `EXTRINSIC` methods maintain equilibrium upon cohesive element insertion.
- `LAMBDA_1` indicates the normalized separation at which the traction response flattens with an additional increase in separation.
- `LAMBDA_2` indicates the normalized separation at which the traction begins to degrade with an additional increase in separation.
- The separation at which failure occurs in the normal direction is prescribed using the `NORMAL LENGTH SCALE` command.
- The maximum traction is specified through the `PEAK TRACTION` command.
- The separation at which failure occurs in the tangential direction is prescribed using the `TANGENTIAL LENGTH SCALE` command.
- The `PENETRATION STIFFNESS MULTIPLIER` command artificially increases the normal traction to help prevent interpenetration of cohesive surfaces when crack closure occurs. However, cohesive elements are not equipped to handle compression, and this penalty stiffness is simply an ad hoc approach to contact.
- Set the `USE ELASTIC UNLOADING` command to `YES` to force this model to unload elastically.

The state variables for this model are listed in Table 5.35.

Table 5.35: State Variables for TVERGAARD HUTCHINSON Surface Model (Section 5.3.2)

Index	Name	Variable Description
0	PEAK_TRACTION	maximum traction the model can experience
1	LAMBDA_MAX	maximum lambda the model has experienced
2	TRACTION_AT_LAMBDA_MAX	traction at LAMBDA_MAX

5.3.3 Thouless Parmigiani

```
BEGIN PARAMETERS FOR MODEL THOULESS_PARMIGIANI
  INIT TRACTION METHOD = IGNORE|ADD|EXTRINSIC (IGNORE)
  LAMBDA_1_N = <real>
  LAMBDA_1_T = <real>
  LAMBDA_2_N = <real>
  LAMBDA_2_T = <real>
  NORMAL LENGTH SCALE = <real>
  PEAK NORMAL TRACTION = <real>
  TANGENTIAL LENGTH SCALE = <real>
  PEAK TANGENTIAL TRACTION = <real>
  PENETRATION STIFFNESS MULTIPLIER = <real>
  USE ELASTIC UNLOADING = NO|YES (YES)
END [PARAMETERS FOR MODEL THOULESS_PARMIGIANI]
```

The Thouless Parmigiani model supports mixed-mode fracture more accurately than the Tvergaard Hutchinson model by separating the normal and tangential components, allowing one to fail independently of the other. Failure of this model is dependent on the energy release of both normal and tangential components. The shape of the traction-separation curve for this model is hardening, followed by a plateau, followed by softening.

In the above command block:

- For dynamically activated or dynamically inserted cohesive surface elements, an initial traction can be used to initialize the traction-separation model based on element properties specified in the `ELEMENT DEATH` or `XFEM` block. This is set via the `INIT TRACTION METHOD`. The default behavior is to ignore any initial tractions and let the traction-separation law dictate the behavior. Note that this does not maintain equilibrium upon cohesive element insertion. The `ADD` method offsets the traction-separation law by the initiation traction, which is linearly scaled to zero under further loading. The `EXTRINSIC` method initializes the state of the traction-separation law along the hardening branch. In the case that the initialization traction is greater than the specified peak traction, the traction-separation law is evaluated from the beginning of the traction plateau with the peak traction value reset to the initialization traction. Both the `ADD` and `EXTRINSIC` methods maintain equilibrium upon cohesive element insertion.
- `LAMBDA_1_N` indicates the normalized normal separation at which the traction response flattens with an additional increase in separation.
- `LAMBDA_1_T` indicates the normalized tangential separation at which the traction response flattens with an additional increase in separation.
- `LAMBDA_2_N` indicates the normalized normal separation at which the traction begins to decrease with an additional increase in separation.
- `LAMBDA_2_T` indicates the normalized tangential separation at which the traction begins to decrease with an additional increase in separation.
- The separation at which failure occurs in the normal direction is prescribed using the `NORMAL LENGTH SCALE` command.
- The maximum normal traction is specified through the `PEAK NORMAL TRACTION` com-

mand.

- The separation at which failure occurs in the tangential direction is prescribed using the `TANGENTIAL LENGTH SCALE` command.
- The maximum tangential traction is specified through the `PEAK TANGENTIAL TRACTION` command.
- The `PENETRATION STIFFNESS MULTIPLIER` command artificially increases the normal traction to help prevent interpenetration of cohesive surfaces when crack closure occurs. However, cohesive elements are not equipped to handle compression, and this penalty stiffness is simply an ad hoc approach to contact.
- Set the `USE ELASTIC UNLOADING` command to `YES` to force this model to unload elastically.

The state variables for this model are listed in Table 5.36.

Table 5.36: State Variables for THOULESS PARMIGIANI Surface Model (Section 5.3.3)

Index	Name	Variable Description
0	FRACTION_OF_FAILURE	current fraction of failure
1	PEAK_TRACTION_N	maximum normal traction the model can experience
2	PEAK_TRACTION_T	maximum tangential traction the model can experience
3	LAMBDA_MAX_N	maximum normal lambda the model has experienced
4	TRACTION_AT_LAMBDA_MAX_N	normal traction at LAMBDA_MAX_N
5	LAMBDA_MAX_T	maximum tangential lambda the model has experienced
6	TRACTION_AT_LAMBDA_MAX_T	tangential traction at LAMBDA_MAX_T
7	G_AT_LAMBDA_MAX_N	the area under the normal traction separation curve up to LAMBDA_MAX_N
8	G_AT_LAMBDA_MAX_T	the area under the tangential traction separation curve up to LAMBDA_MAX_T

5.3.4 Compliant Joint

```
BEGIN PARAMETERS FOR MODEL COMPLIANT_JOINT
  NORMAL MODULUS = <real>norm_mod
  SHEAR MODULUS = <real>shear_mod
  APERTURE = <real>aperture
  APERTURE LIMIT = <real>aperture_limit
  FRICTION COEFFICIENT = <real>fric_coef
  COHESION = <real>cohesion
END [PARAMETERS FOR MODEL COMPLIANT_JOINT]
```

The compliant joint model is formulated for use in modeling of geologic faults and joints. The joint has an elastic response when both opening and closing are defined by `norm_mod` and `shear_mod`. Normal and shear modulus have units of stress.

When the joint is closing, the modulus of the joint varies from `norm_mod` to infinity over a gap distance of `aperture`. The `aperture_limit` is a number between 0.0 and 1.0 that defines how far along the aperture curve the model will go while still increasing the stiffness. A small value causes the model to have a nearly constant stiffness during joint closing. A larger number causes more stiffening of the joint as the over closure approaches the aperture. A value of 1.0 allows the normal stiffness to go all the way to infinity. Values of aperture limit near one are not recommended as they tend to cause the joint model to become extremely stiff, causing the model to become unstable. The aperture has units of length, while the aperture limit is a dimensionless constant.

When the joint is open, it has a shear stiffness specified by `shear_mod`. When the joint is closing, Coulomb frictional effects are added to the shear force. These are computed as `fric_coef` times the compressive force, where `fric_coef` is a dimensionless friction coefficient.

Additional cohesive resistance to joint opening is provided by `cohesion`. This parameter has units of stress.

5.3.5 Functional Interface Traction

```
BEGIN PARAMETERS FOR MODEL FUNCTIONAL_INTERFACE_TRACTION
  NORMAL LENGTH SCALE = <real> (1.0)
  TANGENTIAL LENGTH SCALE = <real> (1.0)
  NORMAL TRACTION FUNCTION = <string>
  TANGENTIAL TRACTION FUNCTION = <string>
END [PARAMETERS FOR MODEL FUNCTIONAL_INTERFACE_TRACTION]
```

The Functional Interface Traction model allows a general normal and tangential traction response law. The normal and tangential components of the traction are defined as functions of the normal and tangential components, respectively, of the separation displacement (appropriately normalized). The traction is evaluated given a separation displacement and subsequently applied. (Note: This model does not presently support some of the features of the other cohesive zone material models, such as elastic unloading and failure.)

The command block for the Functional Interface Traction model starts with the line:

```
BEGIN PARAMETERS FOR MODEL FUNCTIONAL_INTERFACE_TRACTION
```

and terminates with the line:

```
END [PARAMETERS FOR MODEL FUNCTIONAL_INTERFACE_TRACTION]
```

In the above command block:

- The density of the material is defined with the `DENSITY` command line. Although this is not used in the model, it is currently a required input parameter.
- The separation scale in the normal direction is specified using the `NORMAL LENGTH SCALE` command. This is not strictly necessary as this scaling may instead be effected directly within the normal traction response function. It is only provided for similarity with the other cohesive zone material models.
- The separation scale in the tangential direction is specified using the `TANGENTIAL LENGTH SCALE` command. This is not strictly necessary as this scaling may instead be effected directly within the tangential traction response function. It is only provided for similarity with the other cohesive zone material models.
- The normal traction response as a function of the (normalized) normal component of the separation displacement is specified through the `NORMAL TRACTION FUNCTION` command.
- The tangential traction response as a function of the (normalized) tangential component of the separation displacement is specified through the `TANGENTIAL TRACTION FUNCTION` command.

5.3.6 Alves-Roehl

```
begin parameters for model quadshell_model_alves_roehl
  NORMAL LENGTH SCALE = <real>
  TANGENTIAL LENGTH SCALE 1 = <real>
  TANGENTIAL LENGTH SCALE 2 = <real>
  ROTATIONAL LENGTH SCALE = <real>

  # normal work of separation
  NORMAL WOS = <real>

  # tangential work of separation
  TANGENTIAL WOS 1 = <real>
  TANGENTIAL WOS 2 = <real>

  # rotational work of separation
  ROTATIONAL WOS = <real>

  USE ELASTIC UNLOADING = NO|YES (YES)
end
```

In the above command block:

- The separation at which failure occurs in the normal direction is prescribed using the `NORMAL LENGTH SCALE` command.
- The separation scale in the tangential direction 1 is specified using the `TANGENTIAL LENGTH SCALE 1` command.
- The separation scale in the tangential direction 2 is specified using the `TANGENTIAL LENGTH SCALE 2` command.
- The characteristic length for rotational bending is prescribed using the `ROTATIONAL LENGTH SCALE` command.
- The normal work of separation (WOS) in the normal direction is set using the `NORMAL WOS` command.
- The tangential work of separation in the tangential direction 1 is set using the `TANGENTIAL WOS 1` command.
- The tangential work of separation in the tangential direction 2 is set using the `TANGENTIAL WOS 2` command.
- The rotational work of separation (WOS) is set using the `ROTATIONAL WOS` command.
- Set the `USE ELASTIC UNLOADING` command to `YES` to force this model to unload elastically.

References

- [1] W.M. Scherzinger and D.C. Hammerand. Constitutive models in LAME. Technical Report SAND2007-5873, Sandia National Laboratories, Albuquerque, NM, September 2007. [pdf](#).
- [2] A. E. H. Love. *A Treatise on the Mathematical Theory of Elasticity*. Dover, 4th edition, 1944.

- [3] I. S. Sokolnikoff. *Mathematical Theory of Elasticity*. Krieger Publishing Company, 1956.
- [4] S. Timoshenko and J.N. Goodier. *Theory of Elasticity*. McGraw Hill Book Company, 3rd edition, 1970.
- [5] C.M. Stone. SANTOS – a two-dimensional finite element program for the quasistatic, large deformation, inelastic response of solids. Technical Report SAND90-0543, Sandia National Laboratories, Albuquerque, NM, 1996. [pdf](#).
- [6] J.C. Simo and T.J.R. Hughes. *Computational Inelasticity*. Springer-Verlag, New York, NY, 1998.
- [7] C. M. Stone, G. W. Wellman, and R. D. Krieg. A vectorized elastic/plastic power law hardening material model including Lüders strains. Technical Report SAND90-0153, Sandia National Laboratories, Albuquerque, NM, March 1990. [pdf](#).
- [8] G. W. Wellman. A simple approach to modeling ductile failure. Technical Report SAND2012-1343, Sandia National Laboratories, Albuquerque, NM, June 2012. [pdf](#).
- [9] W.F. Hosford. A generalized isotropic yield criterion. *Journal of Applied Mechanics*, 39: 607–609, 1972.
- [10] A. Graf and W. Hosford. Calculations of forming limit diagrams. *Metallurgical Transactions A*, 21A:87–94, 1990.
- [11] W. M. Scherzinger. A return mapping algorithm for isotropic and anisotropic plasticity models using a line search method. *Computer Methods in Applied Mechanics and Engineering*, 317:526–553, 2017.
- [12] R. Hill. *The Mathematical Theory of Plasticity*. Clarendon Press, Oxford, 1950.
- [13] F. Barlat, H. Aretz, J. W. Yoon, J. C. Brem, and R. E. Dick. Linear transformation-based anisotropic yield functions. *International Journal of Plasticity*, 21:1009–1039, 2005.
- [14] G.R. Johnson and W.H. Cook. Fracture characteristics of three metals subjected to various strains, strain rates, temperatures and pressures. *Engineering Fracture Mechanics*, 21(1): 31–48, 1985.
- [15] G.R. Johnson and W.H. Cook. A constitutive model and data for metals subjected to large strains, high strain rates and high temperatures. In *Proc. 7th. Int. Symp. on Ballistics*, pages 541–547, The Hague, The Netherlands, 1983.
- [16] D.J. Bammann, M.L. Chiesa, and G.C. Johnson. Modelling large deformation and failure in manufacturing processes. In T. Tatsumi, E. Watanabe, and T. Kambe, editors, *Proceedings of the 19th International Congress of Theoretical and Applied Mechanics*, pages 359–376, Amsterdam, 1997. Elsevier Science Publishers.
- [17] D.J. Bammann, M.L. Chiesa, M.F. Horstemeyer, and L.E. Weingarten. Failure in ductile materials using finite element methods. In N. Jones and T. Wierzbicki, editors, *Structural Crashworthiness and Failure*, pages 1–53, London, 1993. Elsevier Applied Science.

- [18] D.J. Bammann. Modeling temperature and strain dependent large deformations in metals. *Applied Mechanics Reviews*, 43(5):312–319, 1990. [doi](#).
- [19] A.A. Brown and D.J. Bammann. Validation of a model for static and dynamic recrystallization in metals. *International Journal of Plasticity*, 32–33:17–35, 2012.
- [20] U.F. Kocks and H. Mecking. A mechanism for static and dynamic recovery. In P. Haasen, V. Gerold, and G. Kostorz, editors, *Strength of Metals and Alloys*, pages 345–350, Oxford, 1979. Pergamon Press.
- [21] E. Nes. Recovery revisited. *Acta Metall. Mater.*, 43:2189–2207, 1990.
- [22] S. Kok, A.J. Beaudoin, and D.A. Tortorelli. On the development of stage IV hardening using a model based on the mechanical threshold. *Acta Mater.*, 50:1653–1667, 2002.
- [23] A. Cocks and M. Ashby. Intergranular fracture during power-law creep under multiaxial stresses. *Metall. Sci.*, 14:395–402, 1972.
- [24] J. Lemaitre and J.-L. Chaboche. *Mechanics of Solid Materials*. Cambridge University Press, 1990.
- [25] F.K.G. Odqvist. *Mathematical theory of creep and creep rupture*. Clarendon Press, Oxford, 1966.
- [26] M.K. Neilsen, S. N. Burchett, C. M. Stone, and J. J. Stephens. A viscoplastic theory for braze alloys. Technical Report SAND96-0984, Sandia National Laboratories, Albuquerque, NM, 1996. [pdf](#).
- [27] L.M. Taylor and D.P. Flanagan. Pronto3D: A three-dimensional transient solid dynamics program. Technical Report SAND87-1912, Sandia National Laboratories, Albuquerque, NM, March 1989. [pdf](#).
- [28] R.D. Krieg. A simple constitutive description for cellular concrete. Technical Report SAND SC-DR-72-0883, Sandia National Laboratories, Albuquerque, NM, 1978. [pdf](#).
- [29] D.V. Swenson and L. M. Taylor. A finite element model for the analysis of tailored pulse stimulation of boreholes. *International Journal for Numerical and Analytical Methods in Geomechanics*, 7:469–484, 1983. [doi](#).
- [30] M. K. Neilsen, W. Y. Lu, B. Olsson, and T. Hinnerichs. A viscoplastic constitutive model for polyurethane foams. In *Proceedings ASME 2006 International Mechanical Engineering Congress and Exposition*, Chicago, IL, 2006. ASME.
- [31] L.J. Gibson and M.F. Ashby. *Cellular Solids: Structure and Properties*. Cambridge Solid State Science Series. Cambridge University Press, Cambridge, UK, second edition, 1997.
- [32] M.K. Neilsen, H.S. Morgan, and R.D. Krieg. A phenomenological constitutive model for low density polyurethane foams. Technical Report SAND86-2927, Sandia National Laboratories, Albuquerque, NM, April 1987. [pdf](#).

- [33] B. Störakers. On material representation and constitutive branching in finite compressible elasticity. *Journal of the Mechanics and Physics of Solids*, 34:125–145, 1986.
- [34] R. W. Ogden. *Non-Linear Elastic Deformations*. Dover, New York, 1984.
- [35] P. J. Blatz and W. L. Ko. Application of finite elasticity theory to the deformation of rubbery materials. *Transactions of the Society of Rheology*, 6:223–251, 1962.
- [36] G. Bolzon and R. Vitaliani. The blatz-ko model and homogenization. *Archive of Applied Mechanics*, 63:228–241, 1993.
- [37] A.E. Green and W. Zerna. *Theoretical Elasticity, 2nd Edition*. Clarendon Press, Oxford, 1968.
- [38] S.A. English. A 3D orthotropic strain-rate dependent elastic damage material model. Technical Report SAND2014-17469, Sandia National Laboratories, Albuquerque, NM, 2014. [pdf](#).
- [39] M.K. Neilsen, J.D. Pierce, and R.D. Krieg. A constitutive model for layered wire mesh and aramid cloth fabric. Technical Report SAND91-2850, Sandia National Laboratories, Albuquerque, NM, 1993. [pdf](#).
- [40] R.G. Whirley, B.E. Engelmann, and J.O. Halquist. DYNA3D user’s manual, 1991.
- [41] Hibbitt, Karlsson and Sorensen. ABAQUS version 6.6, theory manual, 2006.
- [42] D.C. Hammerand. ABAQUS style finite strain viscoelasticity in Adagio. Technical Report Memo, Sandia National Laboratories, Albuquerque, NM, March 2003.
- [43] D.C. Hammerand. Finite strain viscoelasticity in Adagio and ABAQUS. Technical Report Memo, Sandia National Laboratories, Albuquerque, NM, July 2003.

Chapter 6

Elements

This chapter explains how material, geometric, and other properties are associated with the various element blocks in a mesh file. A mesh file contains, for the most part, only topological information about elements. For example, there may be a group of elements in the mesh file that consists of four nodes defining a planar facet in three-dimensional space. Whether or not these elements are used as shells or membranes in a model of an object is determined by command lines in the input file. The specifics of a material type associated with these four node facets are also set in the input file.

Most elements can be used in either explicit or implicit calculations. If an element is available for one but not the other, this information will be noted for the element. In particular, there are special element implementations available with explicit calculations for Peridynamics and for smoothed particle hydrodynamics (SPH). This chapter also includes descriptions of the commands for mass property calculations, element death, and mesh rebalancing (mesh rebalancing is available for explicit calculations only). “Element-like” capabilities are also discussed in Chapter 6.

Highlights of chapter contents follow. Section 6.1 discusses the `FINITE ELEMENT MODEL` command block, which provides the description of a mesh associated with the elements. Section 6.2 presents the section command blocks that are used to define the different element sections. Next in Section 6.3 are descriptions of command blocks that exhibit element-like functionality. Section 6.3.1 explains the mechanism (explicit only). Section 6.4 describes the `MASS PROPERTIES` command block, which lets the user compute the total mass of the model or the mass of sub-parts of the model once the element blocks are completely defined in terms of geometry and material. Section 6.5 details the `ELEMENT DEATH` command block, which lets the user delete (kill) elements based on various criteria during an analysis. Section 6.6 describes particle embedding (explicit only), which specifies element blocks in which each element is embedded with particles during initialization. A command block for derived quantities that are to be used with transfers or error estimation is discussed in Section 6.7. Section 6.8 presents various options available in explicit calculations for rebalancing of the initial mesh decomposition for parallel runs, which may improve the parallel performance of some problems. Finally, Section 6.9 describes the procedure for adaptively remeshing a portion of the model.

Most of the command blocks and command lines described next appear within the SIERRA scope. There are some exceptions, and these exceptions are noted.

6.1 Finite Element Model

```
BEGIN FINITE ELEMENT MODEL <string>mesh_descriptor
  DATABASE NAME = <string>mesh_file_name
  DATABASE TYPE = <string>database_type(exodusII)
  ALIAS <string>mesh_identifier AS <string>user_name
  OMIT BLOCK <string>block_list
  COMPONENT SEPARATOR CHARACTER = <string>separator
  TIME SCALE FACTOR = <real>timeScaleFactor

  BEGIN BLOCK DEFAULTS
    #
    # Set default values for each 'PARAMETERS FOR
    # BLOCK' command
    #
  END

  BEGIN PARAMETERS FOR BLOCK [<string list>block_names]
    #
    # Command lines that define attributes for
    # a particular element block appear in this
    # command block.
    #
  END [PARAMETERS FOR BLOCK <string list>block_names]
END [FINITE ELEMENT MODEL <string>mesh_descriptor]
```

The input file must point to a mesh file that is to be used for an analysis. The name of the mesh file appears within a `FINITE ELEMENT MODEL` command block, which appears in the SIERRA scope. In this command block, the particular mesh file that describes the model is identified. There will also be one or more `PARAMETERS FOR BLOCK` command blocks in this block. (All the `PARAMETERS FOR BLOCK` command blocks are embedded in the `FINITE ELEMENT MODEL` command block.) A material type and model, a section, and various other parameters for the element block are set within the `PARAMETERS FOR BLOCK` command block. The concept of “section” is explained in Section 6.1.6.

The following element types are supported in Sierra/SM analyses:

Eight-node, uniform-gradient hexahedron Both a midpoint-increment formulation [1] and a strongly objective formulation are implemented [2]. These elements can be used with any of the material models described in Chapter 5.

Six-node, uniform-gradient wedge Both a midpoint-increment formulation [1] and a strongly objective formulation are implemented [2]. These elements can be used with any of the material models described in Chapter 5.

Five-node, uniform-gradient pyramid Both a midpoint-increment formulation [1] and a strongly objective formulation [2] are implemented. These elements can be used with any of the material models described in Chapter 5.

Eight-node, selective-deviatoric hexahedron Only a strongly objective formulation is provided. This element can be used with any of the material models described in Chapter 5. The selective-deviatoric integration rule is specified in the `BEGIN SOLID SECTION` command block using the `FORMULATION` command line as described in Section 6.2.1.

Four-node tetrahedron There is now the regular element formulation for the four-node tetrahedron and a node-based formulation for the four-node tetrahedron. For the regular element formulation, only a strongly objective formulation is implemented. The concept of a node-based four-node tetrahedron is described in [3]. The regular four-node tetrahedron can be used with any of the material models described in Chapter 5. The node-based tetrahedron can be used with any of the material models described in Chapter 5. When using node-based tetrahedron it is important that nodal quantities are used where other element types use element quantities. For example, element variable stress is evaluated with a regular four-node tetrahedron, but in node-based tetrahedron one should use nodal variable `element_stress_##`, where `##` is a one-based integer > 0 . Nodal variables for node-based tetrahedra will exist on all nodes but are zero where they are not used. Element variables for node-based elements serve as intermediate variables and should not be used in post-processing in the same way as other regular element variables.

Ten-node, uniform-gradient tetrahedron Only a strongly objective formulation is implemented. This element can be used with any of the material models described in Chapter 5.

Ten-node, fully-integrated tetrahedron Only a strongly objective formulation is implemented. This element can be used with any of the material models described in Chapter 5. The full integration rule is specified in the `BEGIN SOLID SECTION` command block using the `FORMULATION` command line as described in Section 6.2.1.

Four-node, quadrilateral, uniform-gradient membrane Both a midpoint-increment formulation and a strongly objective formulation are implemented. This element is derived from the Key-Hoff shell formulation [4]. The strongly objective formulation has not been extensively tested, and it is recommended that the midpoint-increment formulation, which is the default, be used for this element type. These elements can be used with any of the following material models described in Chapter 5:

- Elastic
- Elastic-plastic
- Elastic-plastic power-law hardening
- Multilinear elastic-plastic hardening (no failure)

Four-node, quadrilateral shell This shell uses the Key-Hoff formulation [4]. Both a midpoint-increment formulation and a strongly objective formulation are implemented. The strongly objective formulation has not been extensively tested, and it is recommended that the midpoint-increment formulation, which is the default, be used for this element type. These elements can be used with any of the following material models described in Chapter 5:

- Elastic
- Elastic-plastic
- Elastic-plastic power-law hardening

- Multilinear elastic-plastic hardening without failure
- Multilinear elastic-plastic hardening with failure

Four-node, quadrilateral, selective-deviatoric membrane Only a midpoint-increment formulation is implemented. These elements can be used with any of the following material models described in Chapter 5:

- Elastic
- Elastic-plastic
- Elastic-plastic power-law hardening
- Multilinear elastic-plastic hardening (no failure)

Linear elastic shell The linear elastic shell element is linear in both a material and geometric sense. The linear elastic shell element can be used with any material, however it will use only the density and elastic material constants of that material. Use of the linear elastic shell element is specified with the `FORMULATION = NQUAD` command in the section command block.

Three-node, triangular shell  *Explicit* This shell uses the same formulation as the three-node triangular shell in Pronto [1]. A midpoint-increment formulation is implemented. This element can be used with any of the following material models described in Chapter 5:

- Elastic
- Elastic-plastic
- Elastic-plastic power-law hardening
- Multilinear elastic-plastic hardening without failure
- Multilinear elastic-plastic hardening with failure

Two-node beam The beam element is a uniform result model. Strains and stresses are computed only at the midpoint of the element. These midpoint values determine the forces and moments for the beam. The beam element is based on an incremental kinematic formulation that is accurate for large strains and rotations (i.e., this element exactly agrees with a logarithmic strain formulation under large strain axial loading). Thinning of the cross section is taken into account through a constant volume assumption. There are several different sections currently implemented for the beam including rod, tube, bar, box, and ‘I.’ This element can be used with any of the following material models described in Chapter 5:

- Elastic
- Elastic-plastic

Two-node truss The two-node truss element carries only a uniform axial stress. Currently, there is a linear-elastic material model for the truss element.

Two-node spring The two-node spring element computes a uniaxial resistance force based on a nonlinear force-engineering strain function. This element can handle preloads, mass per unit length, resetting of the initial length after preload and any arbitrary loading function.

Two-node damper  *Explicit* The two-node damping element computes a damping force based on the relative velocity of the two nodes along the axis of the element. This element uses only a damping parameter for a material property.

Point mass The point mass element allows the user to put a specified mass and/or rotational inertia at a single node. This element requires input for density, but does not make use of any other material properties.

Particle  *Explicit* The particle element is a zero-dimensional, sphere-topology element which may be used with any of the material models described in Chapter 5.

The command block to describe a mesh file begins with

```
BEGIN FINITE ELEMENT MODEL <string>mesh_descriptor
```

and is terminated with

```
END [FINITE ELEMENT MODEL <string>mesh_descriptor]
```

where `mesh_descriptor` is a user-selected name for the mesh. This section discusses the command lines within the scope of the `FINITE ELEMENT MODEL` command block but outside the scope of the `PARAMETERS FOR BLOCK` command block, as well as the `PARAMETERS FOR BLOCK` command block and the associated command lines for this particular block.

6.1.1 Identification of Mesh File

Nested within the `FINITE ELEMENT MODEL` command block are two command lines (`DATABASE NAME` and `DATABASE TYPE`) that give the mesh name and define the type for the mesh file, respectively. The command line

```
DATABASE NAME = <string>mesh_file_name
```

gives the name of the mesh file with the string `mesh_file_name`. If the current mesh file is in the default directory and is named `job.g`, then this command line would appear as:

```
DATABASE NAME = job.g
```

If the mesh file is in some other directory, the command line would have to show the path to that directory. For parallel runs, the string `mesh_file_name` is the base name for the spread of parallel mesh files. For example, for a four-processor run, the actual mesh files associated with a base name of `job.g` would be `job.g.4.0`, `job.g.4.1`, `job.g.4.2`, and `job.g.4.3`. The database name on the command line would be `job.g`.

Two meta-characters can appear in the name of the mesh file. If the `%P` character is found in the name, it will be replaced with the number processors being used for the run. For example, a job running on 1024 processors has the name `mesh-%P/job.g`, then the name would be expanded to `mesh-1024/job.g` and the associated mesh files would be `mesh-1024/job.g.1024.0000` to `mesh-1024/job.g.1024.1023`. The other recognized meta-character is `%B` which is replaced with the base name of the input file containing the input commands. For example, if the commands are in the file `my_analysis_run.i` and the mesh database name is specified as `%B.g`, then the mesh would be read from the file `my_analysis_run.g`.

If the mesh file does not use the Exodus II format, the mesh file format must be specified using the command line

```
DATABASE TYPE = <string>database_type(exodusII)
```

Currently, only the Exodus II database format is supported by Presto and Adagio for mesh input. Other options may be added in the future.

6.1.2 Alias

It is possible to associate a user-defined name with some mesh entity. The mesh entity names for Exodus II entities are typically the concatenation of the entity type (for example, “block”, “nodelist”, or “surface”), an underscore (“_”), and the entity id. This generated name can be aliased to a more descriptive name by using the `ALIAS` command line:

```
ALIAS <string>mesh_identifier AS <string>user_name
```

This alias can then be used in other locations in the input file in place of the Exodus II name. Examples of this association are as follows:

```
Alias block_1    as Case
Alias block_10   as Fin
Alias block_12   as Nose
Alias surface_1  as Nose_Case_Interface
Alias surface_2  as OuterBoundary
```

The above examples use the Exodus II naming convention described in Section 1.6. Note that the alias may not be recognized by all code capabilities, such as contact block skinning.

6.1.3 Omit Block

If the finite element mesh contains element blocks that should be omitted from the finite element analysis, the `OMIT BLOCK` line command is used:

```
OMIT BLOCK <string>block_list
```

The element blocks listed in the command are removed from the model. Any node sets or surfaces only existing on nodes or elements in the omitted element blocks are also omitted. When using this command, it is necessary to deactivate (comment out) or remove any references to the omitted blocks in the rest of the input file. Note that if this command is used in a parallel analysis, it is possible for the resulting model to become unbalanced if, for example, the omitted element blocks make up a large portion of the elements on one or more processors. In this case with explicit calculations, the mesh can be rebalanced using the `REBALANCE` command described in Section 6.8.1.

If the block is given an alias, then that alias name must be provided to the block omission command; otherwise, the standard entity name, “block_” followed by the entity id, must be used. See Section 1.6 for further discussion of the Exodus II block naming convention.

Examples of omitting element blocks are:

```
Omit Block block_1 block_2
Omit Block block_10
Omit Block Fin Nose
```

6.1.4 Time Scale Factor

The `TIME SCALE FACTOR` command is used to put a uniform scale factor on the time stamps in the input mesh database. This command will have an effect on capabilities such as `READ VARIABLE` (Section 7.1.3) that reads boundary condition data from the input mesh.

6.1.5 Component Separator Character

A variable defined on the mesh database can be used as an initial condition, or a prescribed temperature with the `READ VARIABLE` command. If the variable is a vector or a tensor, then the base name of the variable will be separated from the suffixes with a separator character. The default separator character is an underscore, but it can be changed with the `COMPONENT SEPARATOR CHARACTER` command.

```
COMPONENT SEPARATOR CHARACTER = <string>character|NONE
```

For example, the variable `displacement` can have the suffixes `x`, `y`, etc. By default, the base name is separated from the suffixes with an underscore character, yielding `displacement_x`, `displacement_y`, etc., in the mesh file. The underscore can be replaced as the default separator by using the above command line. If the data used the period as the separator, then the command would be

```
COMPONENT SEPARATOR CHARACTER = .
```

In the `displacement` example the components would then appear in the mesh file as `displacement.x`, `displacement.y`, etc. The separator can be eliminated with an empty string or `NONE`.

6.1.6 Element Block Parameters

```
BEGIN PARAMETERS FOR BLOCK [<string list>block_names]
  MATERIAL = <string>material_name
  MODEL = <string>model_name
  INCLUDE ALL BLOCKS
  REMOVE BLOCK = <string list>block_names
  SECTION = <string>section_id
  LINEAR BULK VISCOSITY =
    <real>linear_bulk_viscosity_value(0.06)
  QUADRATIC BULK VISCOSITY =
    <real>quad_bulk_viscosity_value(1.20)
  HOURGLASS STIFFNESS =
    <real>hourglass_stiff_value(0.05)
  HOURGLASS VISCOSITY =
    <real>hourglass_visc_value(solid = 0.0,
      shell/membrane = 0.0)
  MEMBRANE HOURGLASS STIFFNESS =
    <real>memb_hourglass_stiff_value(0.05)
  MEMBRANE HOURGLASS VISCOSITY =
```



```

    <real>memb_hourglass_visc_value(0.0)
BENDING HOURGLASS STIFFNESS =
    <real>bend_hourglass_stiff_value(0.05)
BENDING HOURGLASS VISCOSITY =
    <real>bend_hourglass_visc_value(0.0)
TRANSVERSE SHEAR HOURGLASS STIFFNESS =
    <real>tshr_hourglass_stiff_value(0.05)
TRANSVERSE SHEAR HOURGLASS VISCOSITY =
    <real>tshr_hourglass_visc_value(0.0)
EFFECTIVE MODULI MODEL =
    <string>PRONTO|ELASTIC|PROBED(PRONTO/ELASTIC)
MINIMUM EFFECTIVE DILATATIONAL MODULI RATIO =
    <real>min_eff_mod_ratio(1.0e-3/1.0e-6/0.0)
MINIMUM EFFECTIVE SHEAR MODULI RATIO =
    <real>min_eff_mod_ratio(0.05/1.0e-6/0.0)
ELEMENT NUMERICAL FORMULATION = <string>OLD|NEW(OLD)
ACTIVE FOR PROCEDURE <string>proc_name DURING PERIODS
    <string list>period_names
INACTIVE FOR PROCEDURE <string>proc_name DURING PERIODS
    <string list>period_names

END [PARAMETERS FOR BLOCK <string list>block_names]

```

The finite element model consists of one or more element blocks. Associated with an element block or group of element blocks will be a `PARAMETERS FOR BLOCK` command block, which is also referred to in this document as an *element-block command block*. The basic information about the element blocks (number of elements, topology, connectivity, etc.) is contained in a mesh file. Specific attributes for an element block must be specified in the input file. If for example, a block of eight-node hexahedra is to use the selective-deviatoric versus mean-quadrature formulation, then the selective-deviatoric formulation must be specified in the input file. The element library is listed at the beginning of Section 6.1.

The element-block command block begins with the input line

```
BEGIN PARAMETERS FOR BLOCK [<string list>block_names]
```

and is terminated with the input line

```
END [PARAMETERS FOR BLOCK <string list>block_names]
```

Here, `block_names` is a list of element blocks assigned to the element-block command block. Such a list must be included on the `BEGIN PARAMETERS FOR BLOCK` input line if the `INCLUDE ALL BLOCKS` line command is not used. If the format for the mesh file is Exodus II, the typical form of a `block_name` is `block_integerID`, where `integerID` is the integer identifier for the block. If the element block is 280, the value of `block_name` would be `block_280`. It is also possible to generate an alias identifier for the element block and use this for the `block_name`. If `block_280` is aliased to `AL6061`, then `block_name` becomes `AL6061`.

All element blocks listed on the `PARAMETERS FOR BLOCK` command line—or, alternatively, all element blocks included using the line command `INCLUDE ALL BLOCKS` in conjunction with

REMOVE BLOCK—must share the same mechanics properties. The command line MATERIAL associates a particular material `material_name` with all elements in the block. The SECTION command specifies a particular element formulation for the block. The specified section differentiates between elements with the identical topology but different formulations; for example, a block of four-node quadrilateral topology may specify a membrane or shell formulation. Depending on the element formulation, the SECTION command line may also allow the assignment of a variety of parameters to an element block.

All command lines that can be used in a PARAMETERS FOR BLOCK command block are described in Sections 6.1.6.1 through 6.1.6.9.

6.1.6.1 Material Property

```
MATERIAL = <string>material_name
MODEL    = <string>model_name
```

The material property specification for an element block is done by using the above two command lines. The property specification references both a MATERIAL command block and a material-model command block, which has the general form PARAMETERS FOR MODEL `model_name`. These command blocks are described in Chapter 5. The MATERIAL command block contains all the parameters needed to define a material, and is associated with an element block (PARAMETERS FOR BLOCK command block) by use of the MATERIAL command line. Some of the material parameters inside the property specification are grouped on the basis of material models. A material-model command block is associated with an element block by use of the MODEL command line.

Consider the following example. Suppose there is a MATERIAL command block with a `material_name` of steel. Embedded within this command block for steel is a material-model command block for an elastic model of steel and an elastic-plastic model of steel. Suppose that for the current element block it is desired to use the material steel with the elastic model. Then the element-block command block would contain the input lines

```
MATERIAL = steel
MODEL     = elastic
```

If, on the other hand, the elastic-plastic model is desired, the element-block command block would contain

```
MATERIAL = steel
MODEL     = elastic_plastic
```

It is important to note that not all material types and element types are fully compatible.

6.1.6.2 Include All Blocks

The INCLUDE ALL BLOCKS line command is used to associate all element blocks with the same element parameters (which minimizes input).

```
INCLUDE ALL BLOCKS
```

6.1.6.3 Remove Block

The `REMOVE BLOCK` command line allows the user to delete blocks from the set specified in the `PARAMETERS FOR BLOCK` command block or `INCLUDE ALL BLOCKS` command line(s) through the string list `block_names`.

```
REMOVE BLOCK <string>block_list
```

6.1.6.4 Section

```
SECTION = <string>section_id
```

The section specification for an element-block command block is done by using the above command line. The `section_id` is a string associated with a section command block. The `SECTION` command line must specify the unique identifier or name of a `SECTION` command block defined by the user elsewhere in the input. The chosen section must also be compatible with the topology of the block as specified in the input mesh file. It is within the `SECTION` command block that element formulation-specific properties are given. Currently available section command blocks are

- `SOLID SECTION`,
- `COHESIVE SECTION`,
- `SHELL SECTION`,
- `MEMBRANE SECTION`,
- `BEAM SECTION`,
- `TRUSS SECTION`,
- `SPRING SECTION`,
- `DAMPER SECTION`,
- `POINT MASS SECTION`,
- `PARTICLE SECTION`,
- `PERIDYNAMICS SECTION`, and
- `SUPERELEMENT SECTION`.

Solid sections are only compatible with three-dimensional solid topologies; cohesive, shell, and membrane sections are only compatible with two-dimensional topologies; beam, truss, spring, and damper sections are only compatible with one-dimensional mesh topologies; while point mass, particle, and peridynamics sections are only compatible with sphere topology in the input mesh. Many additional details on the various `SECTION` command blocks are given in [Section 6.2](#).

If no `SECTION` command line is present in a `PARAMETERS FOR BLOCK` command block, the default eight-node hexahedral formulation based on mean quadrature and midpoint-increment strain incrementation is assumed.

Suppose a membrane formulation is desired for a given element block. First, a `MEMBRANE SECTION` command block with a unique name such as `membrane_rubber` must be defined elsewhere in the input file. Inside the desired element-block command block, one would then specify

```
SECTION = membrane_rubber
```

to properly assign the membrane element formulation to the block. Specific section properties, such as the thickness of the membrane, would be described in the `MEMBRANE SECTION` command block and then associated with the current element-block command block.

Only one `SECTION` command line is allowed per element-block command block. Several element-block command blocks may reference the same section command block. For example, in Figure 6.1, the section named `membrane_rubber` appears in two different `PARAMETERS FOR MODEL` command blocks, while the associated `MEMBRANE SECTION` command block is defined only once.

```
BEGIN FINITE ELEMENT MODEL mesh1
.
.
  BEGIN PARAMETERS FOR BLOCK block1
    .
    SECTION membrane_rubber
    .
  END PARAMETERS FOR BLOCK block1
  BEGIN PARAMETERS FOR BLOCK block2
    .
    SECTION membrane_rubber
    .
  END PARAMETERS FOR BLOCK block2
.
.
END FINITE ELEMENT MODEL mesh1

BEGIN MEMBRANE SECTION membrane_rubber
.
.
END MEMBRANE SECTION membrane_rubber
```

Figure 6.1: Association between `SECTION` command lines and a section command block.

6.1.6.5 Linear and Quadratic Bulk Viscosity

```
LINEAR BULK VISCOSITY =
  <real>linear_bulk_viscosity_value(0.06)
QUADRATIC BULK VISCOSITY =
  <real>quad_bulk_viscosity_value(1.20)
```

The linear and quadratic bulk viscosity are set with these two command lines. Consult the documentation for the elements [5] for a description of the bulk viscosity parameters.

6.1.6.6 Hourglass Control

```
HOURLASS STIFFNESS = <real>hourglass_stiff_value(0.05)
HOURLASS VISCOSITY = <real>hourglass_visc_value(0.0)
MEMBRANE HOURLASS STIFFNESS =
```

```

    <real>memb_hourglass_stiff_value(0.05)
MEMBRANE HOURGLASS VISCOSITY =
    <real>memb_glass_visc_value(0.0)
BENDING HOURGLASS STIFFNESS =
    <real>bend_hourglass_stiff_value(0.05)
BENDING HOURGLASS VISCOSITY =
    <real>bend_glass_visc_value(0.0)
TRANSVERSE SHEAR HOURGLASS STIFFNESS =
    <real>tshr_hourglass_stiff_value(0.05)
TRANSVERSE SHEAR HOURGLASS VISCOSITY =
    <real>tshr_glass_visc_value(0.0)
HOURGLASS TRANSITION STRAIN =
    <real>hourglass_transition_strain_value(1.0)
HOURGLASS EXPONENT = <real>hourglass_exponent_value(1.0)

```

These command lines set the hourglass control parameters for elements that use hourglass control. Currently, the included elements are the eight-node, uniform-gradient hexahedral elements; the eight-node and ten-node tetrahedral elements; and the four-node membrane and shell elements. Consult the element documentation [5] for a description of the hourglass parameters.

Hourglass stiffness and viscosity parameters for under integrated elements are set using the `HOURGLASS STIFFNESS` and `HOURGLASS VISCOSITY` commands, respectively. If either of these commands are used for shell elements, they set the hourglass stiffness or viscosity for all three modes (membrane, bending, and transverse shear).

Hourglass parameters for the membrane, bending, and transverse shear modes can be set individually for shell elements. The membrane hourglass stiffness and viscosity can be set with the `MEMBRANE HOURGLASS STIFFNESS` and `MEMBRANE HOURGLASS VISCOSITY` commands. These membrane hourglass commands can also be used with membrane elements. The bending hourglass stiffness and viscosity are set with the `BENDING HOURGLASS STIFFNESS` and `BENDING HOURGLASS VISCOSITY` commands, and transverse shear hourglass stiffness and viscosity are set with the `TRANSVERSE SHEAR HOURGLASS STIFFNESS` and `TRANSVERSE SHEAR HOURGLASS VISCOSITY` commands. All of these commands will override either the default values and any value set in the generic `HOURGLASS STIFFNESS` and/or `HOURGLASS VISCOSITY` commands for the particular mode that is specified.

The hourglass stiffness parameter defaults to 0.05 for all elements using hourglass control. The hourglass viscosity parameter defaults to 0.0 for all elements using hourglass control.

Hourglass resistances are scaled by the material effective modulus. The hourglass resistance can be strongly affected by the method that computes the effective moduli. The command line in Section 6.1.6.7 selects the method for computing the effective moduli.

The `HOURGLASS TRANSITION STRAIN` and `HOURGLASS EXPONENT` commands only apply to the `HYPERELASTIC` hourglass formulation. The hyperelastic hourglass is a nonlinear hourglass control formulation. The nonlinearity becomes prominent once the hourglass strain reaches the `HOURGLASS TRANSITION STRAIN`. The strength of the nonlinearity is controlled by the `HOURGLASS EXPONENT`, where an exponent of 1.0 gives a linear hourglass force-displacement relation and the default of 2.0 results in a force which is linear at small strains, but becomes a cubic

function of hourglass displacement at large strains. See the theory manual for more details.

6.1.6.7 Effective Moduli Model

```
EFFECTIVE MODULI MODEL =  
  <string>PRONTO|ELASTIC|PROBED (PRONTO/ELASTIC)
```

The element-based critical time step and hourglass force computations require a measure of the material moduli, which is given in Sierra/SM by the effective moduli. The CFL condition dictates that the critical time step is directly related to the current material moduli, and the hourglass forces must be appropriately scaled based on the material moduli. For elastic, isotropic material models, the moduli are constant throughout the analysis. However, for nonlinear materials, the moduli are typically computed numerically from the stresses. For models with softening regimes or that approach perfect plasticity, the moduli may be difficult to define, and the way in which they are computed may adversely affect the analysis. Through the `EFFECTIVE MODULI MODEL` command line, Presto provides several methods for the computation of these effective moduli:

PRONTO This method is the default when running an explicit calculation and is identical to the method of computing effective moduli present in the Pronto3D code. This method will compute an approximate tangent modulus based on current material response. The `PRONTO` method includes some bounds on admissible tangent stiffness values to promote stability. The `PRONTO` method can produce poor results in analyses with materials that undergo substantial hardening; the `PROBED` modulus may be a better option for such materials.

PROBED This method calls the material stiffness calculations twice each step to compute a true tangent probe of the material stiffness. It is generally more accurate and robust than the `PRONTO` method, although more expensive, as it requires two rather than one material call per time step. The `PROBED` option may be worth using for models where the material stiffness changes greatly (such as compressible foams). Note the probe modulus is a probe of the instantaneous tangent modulus and may have difficulty taking in to the account of rate-dependent effects such as creep or strain rate hardening.

MATERIAL A handful of material models are able to compute an analytic tangent stiffness. Currently this includes only the `HYPER FOAM` and the `MOONEY RIVLIN` material models. When the analytic result is available it provides the most robust and most efficient estimate of the tangent stiffness. The `PRONTO` and `PROBED` methods will both use the analytic tangent modulus if possible and revert to a numerical tangent calculation for materials that do not compute an analytic tangent modulus.

ELASTIC This method simply uses the initial elastic moduli for the entire analysis. For materials that harden substantially (such as compressible foams) the elastic method could produce moduli late in the analysis that are too low causing stability problems. For materials that have constant stiffness or if the elastic constants are set in a way to be conservative for the entire analysis routine the elastic modulus method may be the most robust. The elastic modulus is the default method when running an implicit calculation.

Table 6.1: Default moduli ratios for the various effective moduli models.

Model	Dilatational ratio	Shear ratio
PRONTO	0.001	0.05
PROBED	1.0E-6	1.0E-6
ELASTIC	1.0	1.0

The `EFFECTIVE MODULI MODEL` command line should be used with caution because it can strongly affect the analysis results.

The commands

- `MINIMUM EFFECTIVE DILATATIONAL MODULI RATIO` and
- `MINIMUM EFFECTIVE SHEAR MODULI RATIO`

define the minimum effective modulus that can be computed for a material with respect to the initial elastic modulus. This value prevents the computed tangent modulus from dropping to close to zero for yielding or softening materials. A larger value for this constant can sometimes promote stability by allowing the effective modulus to track with material stiffness, but not bottom out at such a low value that low stiffness instability can occur. The computed effective dilatational modulus value primarily affects timestep and bulk viscosity calculations (which can affect the speed of elastic wave propagation). The computed effective shear modulus primarily affects hourglass control in the elements. For historical reasons the default value for these parameters depends on the effective moduli model used; the default ratios are listed in Table 6.1.



6.1.6.8 Element Numerical Formulation

`ELEMENT NUMERICAL FORMULATION = <string>OLD|NEW(OLD)`

For calculation of the critical time step, it is necessary to determine a characteristic length for each element. In one dimension, the correct characteristic element length is the distance between the two nodes of the element. In higher dimensions, this length is usually taken to be the minimum distance between any of the nodes in the element. However, some finite element codes, primarily those based on Pronto3D [1], use as a characteristic length an eigenvalue estimate based on work by Flanagan and Belytschko [6]. That characteristic length provides a stable time step, but often is far more conservative than the minimum distance between nodes. For a cubic element with side length equal to 1, and thus also surface area of each face and volume equal to 1, the minimum distance between nodes is 1. However, the eigenvalue estimate is $\frac{1}{\sqrt{3}}$, which is only 58% of the minimum distance. As the length of the element is increased in one direction while keeping surfaces in the lateral direction squares of area 1, the eigenvalue estimate asymptotes to $\frac{1}{\sqrt{2}}$ for very long elements. If the length is decreased, the eigenvalue estimate asymptotes to the minimum distance between nodes for very thin elements. In this case, the eigenvalue estimate is always more conservative than the minimum distance between nodes. However, consider an element whose cross section in one direction is not a square but a trapezoid with one side length much greater than the other. Assume the large side length is 1 and the other side length is arbitrarily small, ε . In this case, the minimum distance between nodes becomes ε , creating a small and inefficient time step. However,

the eigenvalue estimate is related to the length across the middle of the trapezoid, which for the conditions stated is $\frac{1}{2}$. Since both distances provide stable time steps, and one or the other can be much larger in various circumstances, the most efficient calculation is obtained by using the maximum of the two lengths, either the eigenvalue estimate or the minimum distance between nodes, to determine the time step.

By using the maximum of the lengths, the computed critical time step will be very nearly unstable, and the `TIME STEP SCALE FACTOR` command line should be used to provide a margin of safety; the scale factor should not be greater than 0.9, and sometimes it may need to be reduced further. Although this time step estimate is closer to the critical value and provides better accuracy and efficiency, a smaller scale factor may need to be specified for stability. For this reason, the choice of approach to use is left to the user and is determined by the command line

```
ELEMENT NUMERICAL FORMULATION = <string>OLD|NEW(OLD)
```

If the input parameter is `OLD`, only the eigenvalue estimate is used; `NEW` means that the maximum of the two lengths is used. The default is `OLD` so that users will have to specifically choose the new approach and be aware of the scale factor for the time step.

The `ELEMENT NUMERICAL FORMULATION` command line is applicable to both the energy-dependent and purely mechanical material models. If this command line is applied to blocks using energy-dependent materials, only the determination of the characteristic length is affected. If this command line is applied to an element block with a purely mechanical model and the `OLD` option is used, the Pronto3D-based artificial viscosity, time step, and eigenvalue estimate will be used in the element calculations. If, however, the `NEW` option is used, the artificial viscosity and time step will be computed from equations associated with the energy-dependent models. Further details about the critical time step calculations and the use of this command line are available in [7].

6.1.6.9 Activation/Deactivation of Element Blocks by Time

```
ACTIVE FOR PROCEDURE <string>proc_name DURING PERIODS
    <string list>period_names
INACTIVE FOR PROCEDURE <string>proc_name DURING PERIODS
    <string list>period_names
```

This command line permits the activation and deactivation of element blocks by time period. The time periods are defined in the `TIME STEPPING BLOCK` command block (Section 3.1.1) within a specific procedure named in a `PRESTO PROCEDURE` or an `ADAGIO PROCEDURE` command block (Section 2.2.1).

The `ACTIVE FOR PROCEDURE` or `INACTIVE FOR PROCEDURE` command lines can optionally be used to deactivate element blocks for a portion of the analysis. If the `ACTIVE FOR PROCEDURE` command is used, the element block is active for all periods listed for the named procedure, and is deactivated for all time periods that are absent from the list. If the `INACTIVE FOR PROCEDURE` command is used, the element block is deactivated for all periods listed for the named procedure. The element block is active for all time periods that are absent from the list. If neither command line is used, by default the block is active during all time periods. This command line controls the activation and deactivation of all elements in a block. Alternatively, individual elements can be deactivated with the `ELEMENT DEATH` command block (see Section 6.5).



Warning: Inactive element's corresponding nodes will not move while they are inactive except when the nodes are shared with active elements. It is therefore possible that inactive elements can be inverted while inactive and trigger a fatal error when activated.



Known Issue: Deactivation of element blocks does not currently work in conjunction with the full tangent preconditioner (see Section 4.3) in Adagio. To use this capability, one of the nodal preconditioners must be used.



Known Issue: Deactivating a block only turns off the element/material computations and does not remove the blocks degrees of freedom from the solution matrix. Thus if the nodes have some external force applied convergence may be impossible in implicit or the nodes may accelerate to infinite velocity in explicit. It is recommended that on time periods that a block is deactivated a kinematic boundary condition that fixes all degrees of freedom of the deactivated block also be used.

6.1.7 Element Block Default Parameters

```
BEGIN BLOCK DEFAULTS
  MATERIAL = <string>material_name
  MODEL     = <string>model_name
  SECTION   = <string>section_id
  LINEAR BULK VISCOSITY =
    <real>linear_bulk_viscosity_value(0.06)
  QUADRATIC BULK VISCOSITY =
    <real>quad_bulk_viscosity_value(1.20)
  HOURGLASS STIFFNESS =
    <real>hourglass_stiff_value(0.05)
  HOURGLASS VISCOSITY =
    <real>hourglass_visc_value(0.0)
  EFFECTIVE MODULI MODEL =
    <string>PRONTO|ELASTIC|PROBED(PRONTO/ELASTIC)
  MINIMUM EFFECTIVE SHEAR MODULI RATIO =
    <real>min_eff_mod_ratio
  MINIMUM EFFECTIVE DILATATIONAL MODULI RATIO =
    <real>min_eff_mod_ratio
END [PARAMETERS FOR BLOCK <string list>block_names]
```

The block defaults block may be used modify the default values for commands used inside the BEGIN PARAMETERS FOR BLOCK blocks within the same BEGIN FINITE ELEMENT MODEL command block. See section 6.1.6 for the description of each command.

Block defaults can be used to simplify input when a model has a large number of blocks. For example to try an alternative effective moduli method or hourglass stiffness at all blocks at once. If a given parameter is set in both block defaults and a specific block the specific block value

overrides the block default value. The following examples demonstrate use of the block defaults command.

Example 1:

```
begin block defaults
  hourglass stiffness = 0.10
  hourglass viscosity = 0.01
end
begin parameters for block block_1
  material = steel
  model    = elastic
end
begin parameters for block block_2
  material = rubber
  model    = mooney_rivlin
end
```

In this example the hourglass stiffness and viscosity defaults have been changed for all blocks. This would be equivalent to the following input:

```
begin parameters for block block_1
  material = steel
  model    = elastic
  hourglass stiffness = 0.10
  hourglass viscosity = 0.01
end
begin parameters for block block_2
  material = rubber
  model    = mooney_rivlin
  hourglass stiffness = 0.10
  hourglass viscosity = 0.01
end
```

Example 2:

```
begin block defaults
  material = aluminum
  model    = elastic_plastic
  section  = solid_1
end
begin parameters for block block_1
end
begin parameters for block block_2
end
begin parameters for block block_3
```

```

end
begin parameters for block block_4
  material = tungsten
  model    = elastic
end

```

A default material is used for blocks one two and three, but the defaults are partially overridden for block 4. This would be equivalent to the following input:

```

begin parameters for block block_1 block_2 block_3
  material = aluminum
  model    = elastic_plastic
  section  = solid_1
end
begin parameters for block block_4
  material = tungsten
  model    = elastic
  section  = solid_1
end

```

6.2 Element Sections

Element sections are defined by section command blocks. There are currently nine different types of section command blocks. The section command blocks appear in the SIERRA scope, at the same level as the `FINITE ELEMENT MODEL` command block. In general, a section command block has the following form:

```
BEGIN section_type SECTION <string>section_name
  command lines dependent on section type
END [section_type SECTION <string>section_name]
```

Currently, `section_type` can be `SOLID`, `COHESIVE`, `SHELL`, `MEMBRANE`, `BEAM`, `TRUSS`, `SPRING`, `DAMPER`, `POINT MASS`, `PARTICLE`, `PERIDYNAMICS`, or `SUPERELEMENT`. These various section types are identified as individual section command blocks and are described below. The corresponding `section_name` parameter in each of these command blocks, e.g., `truss_section_name` in the `TRUSS SECTION` command block, is selected by the user. The method used to associate these names with individual `SECTION` command lines in `PARAMETERS FOR BLOCK` command blocks is discussed in Section 6.1.6.4.

Following is a list of most of the different element formulations that are offered in Sierra/Solid-Mechanics. The sections are colored red, green, and blue based on where the documentation can be found. Red sections do not have any documentation posted; prospective users should contact a developer before use. Green sections can be found in the Capabilities in Development manual. Blue links to the relevant section in this manual.

Element	# nodes	Section	Formulation
ANY	ANY	RIGID BODY	
		SUPER ELEMENT	
Sphere	1	DEM	
		PARTICLE	SPH
		PERIDYNAMICS	POINT MASS
		RKPM	
BAR	2	BEAM	
		BOLT	
		CYLINDRICAL JOINT	
		DAMPER	
		LINEAR BEAM	
		SPRING	
		SPRING SUPPORT	
		TRUSS	
		USERMOUNT	

Element	# nodes	Section	Formulation	Strain Incrementation
Shell	3	SHELL	ORIG_TRI_SHELL	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
			C0_TRI_SHELL	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
	4	COHESIVE		
		MEMBRANE	MEAN_QUADRATURE MEAN_QUADRATURE	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
			SELECTIVE_DEVIATORIC	MIDPOINT_INCREMENT
		SHELL	FIBER_SHELL	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
			NQUAD	
			SEVEN_PARAMETER	
			BT_SHELL	MIDPOINT_INCREMENT
			KH_SHELL	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
			BT_SC_SHELL	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
			BL_SHELL	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
			HEX_SHELL	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE
TETRA		SOLID	MEAN_QUADRATURE	STRONGLY_OBJECTIVE NODE_BASED
			FULLY_INTEGRATED	STRONGLY_OBJECTIVE NODE_BASED
			VOID	
PYRAMID	5	SOLID	MEAN_QUADRATURE	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE TOTAL_LAGRANGE
WEDGE	6	COHESIVE		
		LOCAL		
		SOLID	MEAN_QUADRATURE	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE TOTAL_LAGRANGE
HEX	8	COHESIVE		
		LOCAL		
		SOLID	ENHANCED_STRAIN	
			FULLY_INTEGRATED	
			MEAN_QUADRATURE	MIDPOINT_INCREMENT STRONGLY_OBJECTIVE TOTAL_LAGRANGE
			QIP0	STRONGLY_OBJECTIVE
			SELECTIVE_DEVIATORIC	STRONGLY_OBJECTIVE
			THICKSHELL	
			VOID	
TETRA	8	NOT SUPPORTED		
TETRA	10	SOLID	MEAN_QUADRATURE	STRONGLY_OBJECTIVE
			FULLY_INTEGRATED	
HEX	20	SOLID	FULLY_INTEGRATED	STRONGLY_OBJECTIVE
HEX	27	SOLID	FULLY_INTEGRATED	STRONGLY_OBJECTIVE

TOTAL LAGRANGE SECTION

Element	# nodes	Formulation	Strain Incrementation	cubature
TETRA	4	FULLY_INTEGRATED	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	1
			LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	2
			LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	3
HEX	8	FULLY_INTEGRATED	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	ANY
TETRA	10	COMPOSITE_TET	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	2
			LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	3
		FULLY_INTEGRATED	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	2
			LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	3
			LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	4
PYRAMID	13	FULLY_INTEGRATED	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	ANY
WEDGE	15	FULLY_INTEGRATED	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	ANY
HEX	20	FULLY_INTEGRATED	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	ANY
HEX	27	FULLY_INTEGRATED	LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	3
			LOGARITHMIC_MAPPING STRONGLY_OBJECTIVE	5

6.2.1 Solid Section

```
BEGIN SOLID SECTION <string>solid_section_name
  RVE COORDINATE SYSTEM = <string>Coordinate_system_name
  FORMULATION = <string>MEAN_QUADRATURE|SELECTIVE_DEVIATORIC|
    FULLY_INTEGRATED|VOID|Q1P0 (MEAN_QUADRATURE)
  DEVIATORIC PARAMETER = <real>deviatoric_param
  STRAIN INCREMENTATION = <string>MIDPOINT_INCREMENT|
    STRONGLY_OBJECTIVE|NODE_BASED (depends)
  NODE BASED ALPHA FACTOR = <real>bulk_stress_weight (0.01)
  NODE BASED BETA FACTOR = <real>shear_stress_weight (0.35)
  NODE BASED STABILIZATION METHOD = <string>EFFECTIVE_MODULI|
    MATERIAL (MATERIAL)
  HOURGLASS FORMULATION = <string>TOTAL|INCREMENTAL|
    HYPERELASTIC (INCREMENTAL)
  HOURGLASS INCREMENT = <string>ENDSTEP|MIDSTEP (ENDSTEP)
  HOURGLASS ROTATION = <string> APPROXIMATE|SCALED (APPROXIMATE)
  RIGID BODY = <string>rigid_body_name
  RIGID BODIES FROM ATTRIBUTES = <integer>first_id
    TO <integer>last_id
  Q1P0 STABILIZATION THRESHOLD = <real>threshold (0.0)
  Q1P0 TIMESTEP SCALE FACTOR = <real>scale_factor (0.95)
  Q1P0 TIMESTEP WAVE SPEED = <string>VOLUMETRIC|SHEAR|
    AUTOMATIC (AUTOMATIC)
  Q1P0 TIMESTEP LENGTH SCALE = <string>DEFORMED_NODAL_DISTANCE|
    MINIMUM_MAPPING_STRETCH|INSCRIBED_SPHERE_DIAMETER
    (MINIMUM_MAPPING_STRETCH)
  MASS LUMPING METHOD = DEFAULT|ROWSUM|EVEN (DEFAULT)
END [SOLID SECTION <string>solid_section_name]
```

The `SOLID SECTION` command block is used to specify the properties for solid elements (hexahedra and tetrahedra). This command block is to be referenced by an element block made up of solid elements. The two types of solid-element topologies currently supported are hexahedra and tetrahedra. The parameter `solid_section_name` is user-defined and is referenced by a `SECTION` command line in a `PARAMETERS FOR BLOCK` command block.

The `RVE COORDINATE SYSTEM` command line specifies the name of a coordinate system definition block command used to define a local coordinate system on each element of a block that uses this `SOLID SECTION`. This coordinate system only has an effect on RVE analysis (see the *Capabilities in Development* manual for details).

The `FORMULATION` command line specifies whether the element will use a single-point integration rule (mean quadrature), a selective-deviatoric rule, a selectively integrated rule, or a fully-integrated rule, or act as a void element. The selective-deviatoric and fully-integrated integration rules are higher-order integration schemes, which are discussed below.

If the user wishes to use the selective-deviatoric rule, the `DEVIATORIC PARAMETER` command line must also appear in the `SOLID SECTION` command block.

Choosing the `SELECTIVE_DEVIATORIC` formulation indicates that the element is fully integrated with respect to the deviatoric stress response and under integrated with respect to the hydrostatic pressure response. The under-integration of the hydrostatic pressure response is achieved through averaging the pressure over the integration points (note, however, that this averaging is slightly different from the Q1P0 averaging scheme). The `SELECTIVE_DEVIATORIC` formulation is often used to alleviate volumetric locking associated with isochoric motion. By default, this formulation involves no hourglass control. However, the user may include an additional input command within the element section, the `deviatoric_param`, which may be set between 0.0 and 1.0, and which blends the `MEAN_QUADRATURE` and `SELECTIVE_DEVIATORIC` integration schemes. The `MEAN_QUADRATURE` and `SELECTIVE_DEVIATORIC` schemes are recovered at 0.0 and 1.0, respectively. By default, this parameter is set to 1.0, which is the recommended value.

The selective-deviatoric elements, when used with a value greater than 0.0, provide hourglass control without artificial hourglass parameters.

Alternatively, a fully-integrated rule can be defined by setting `FORMULATION = FULLY_INTEGRATED`. This will apply an eight-point integration scheme for eight-noded hexahedral elements and a four-point integration scheme for ten-noded tetrahedral elements. This definition is currently unavailable for four-noded and eight-noded tetrahedral elements.

A selectively integrated formulation can be specified with the command `FORMULATION = Q1P0`. This is only available for 8-node hexahedral element blocks. Additional details about this element formulation are given in Section 6.2.1.2.

The `STRAIN_INCREMENTATION` command defines the strain formulation. Different element formulations may use midpoint increment, strongly objective, or have the option to use either. The overall simulation sensitivity to different choices of this incrementation is usually slight. Generally `STRONGLY_OBJECTIVE` is slightly more accurate and mildly slower than `MIDPOINT_INCREMENT`.

The `VOID` formulation is valid for 8-node hexahedral and 4-node tetrahedral element blocks. Void elements only compute volume. They do not contribute internal forces to the model. The material model and density associated with void elements are ignored. The volume and the first and second derivatives of the volume for each element are stored in the element variables `volume`, `volume_first_derivative`, and `volume_second_derivative`. The volume derivatives are computed using least squares fits of the volume history, which is stored for the previous five time steps.

In addition to the per-element volume and derivatives, the total volume and derivatives of that total volume for all elements in each void element block are written to the results file as global variables. The names for these variables are `voidvol_blockID`, `voidvol_first_deriv_blockID`, and `voidvol_second_deriv_blockID`. In these global variable names, `blockID` is the ID of the block. For example, the void volume for block 8 would be stored in `voidvol_8`.

Some solid elements support a variety of strain incrementation formulations. The element summary at the beginning of Section 6.1 specifies the strain incrementation formulations available for each element. The `STRAIN_INCREMENTATION` command line lets the user specify a midpoint-increment strain formulation (`MIDPOINT_INCREMENT`), a strongly objective strain formulation (`STRONGLY_OBJECTIVE`), or a node-based formulation (`NODE_BASED`) for some elements (see Section 6.2.1.1). Consult the element documentation [2, 5] for a description of these strain formu-

lations.

The `HOURGLASS FORMULATION` command is used to switch between total, hyperelastic and incremental forms of hourglass control. This option can only be used with eight-noded uniform-gradient hexahedral elements using strongly objective strain incrementation (`STRAIN INCREMENTATION = STRONGLY_OBJECTIVE`). One of the following three arguments can be used with this command: `TOTAL`, `HYPERELASTIC`, or `INCREMENTAL`.

The total formulation performs stiffness hourglass force updates based on the rotation tensor from the polar decomposition of the total deformation gradient. The hyperelastic formulation computes hourglass strains relative to the reference/model configuration, and has a well defined hyperelastic hourglass strain energy; as a result this formulation results in a symmetric contribution to the stiffness matrix, is fully reversible, and allows for a nonlinear hourglass resistance. The incremental formulation is the default and performs stiffness hourglass force updates based on the hourglass velocities and the incremental rotation tensor. The viscous hourglass forces and the hourglass parameters are unchanged by this command. Consult the element documentation [5] for a description of the hourglass forces and the incremental hourglass formulation.

The `HOURGLASS INCREMENT` and `HOURGLASS ROTATION` commands control the speed and accuracy of the hourglass control computation. These commands are only applicable to the uniform gradient hexahedron with midpoint strain incrementation (`STRAIN INCREMENTATION = MIDPOINT_INCREMENT`). The `HOURGLASS INCREMENT` line command specifies whether the hourglass resistance increment is to be computed at the middle or end of the time step. The end-step calculation has a slightly reduced computational cost while the mid-step computation is more accurate. The default is `ENDSTEP`. The `HOURGLASS ROTATION` command controls whether the hourglass resistance will be scaled after rotation to preserve the magnitude. Scaling requires additional computation time but will be more accurate, particularly when large rotations are present in the analysis. The default is `APPROXIMATE`, meaning no scaling is done.

Rigid elements in a section are indicated by including the `RIGID BODY` command line. The `RIGID BODY` command line specifies an identifier that maps to a rigid body command block. See Section 6.3.1 for a full discussion of how to create rigid bodies and Section 6.3.1.1 for information on the use of the `RIGID BODIES FROM ATTRIBUTES` command.



Known Issue: For problems with large rotations the hourglass energies are known to spike exponentially with increases in total rotation. This may be due to a coupling with hypoelastic material models.

6.2.1.1 Node Based Tetrahedra

The node-based formulation can only be used with four node tetrahedral elements. If the element-block (i.e., `PARAMETERS FOR BLOCK`) command block has a `SECTION` command line that references a `SOLID SECTION` command block that uses

```
STRAIN INCREMENTATION = NODE_BASED
```

then the tetrahedral element block will use the node-based tetrahedron formulation.

The node-based formulation calculates a solution that is a mixture of an element-based formulation (information from the center of an element) and a node-based formulation (information at a node that is based on the average of all elements attached to the node). The node-based tetrahedron allows a model with four-node tetrahedral elements to avoid most of the locking problems of regular tetrahedral elements. Regular tetrahedral elements are much too stiff and can produce inaccurate results in many situations.

The mixture of node-based versus element-based information incorporated into the solution may be adjusted with the `NODE BASED ALPHA FACTOR` and `NODE BASED BETA FACTOR` command lines. These two lines apply only if the `NODE_BASED` option is specified on the `STRAIN INCREMENTATION` command line. The value for `bulk_stress_weight` on the `NODE BASED ALPHA FACTOR` command line sets the element bulk stress weighting factor, while the value for `shear_stress_weight` on the `NODE BASED BETA FACTOR` command line sets the element shear stress weighting factor. The weighting factors are described in detail in [3]. A strictly node-based formulation results when both factors are set to zero, while a strictly element-based formulation results when both factors are one.

The mixing of node-based and element-based solutions for the node-based formulation is a stabilization technique to eliminate zero energy modes of a pure node-based formulation. With this in mind, two options are provided for the stress update of the element-based solution through the `NODE BASED STABILIZATION METHOD` command (applicable only with `STRAIN INCREMENTATION = NODE_BASED`). The default option, `MATERIAL`, runs the same material model in the element as is run at the nodes, with independent material states. The other option, `EFFECTIVE_MODULI`, uses a linearized update for the element stress state based on the element average value of the nodal effective moduli. The nodal effective moduli are computed from the nodal stress and strain increments at each time step. This second option avoids potential issues due to having independent node and element material states.

Node based elements store material state, stress, and strain data at the nodes in addition to at the element center. Most element based quantities such as stress have similarly named nodal equivalents.

If a node is attached to only elements of one material type that node will have one set of state, stress and strain data that is the average of the attached element quantities.

If elements of multiple materials are attached to the same node then that node will store multiple independent sets of state variables. To accommodate this the node based state data is given a unique name by appending a unique index number to the end of each state variable name.

The log file contains a table that defines the mapping from the index number the nodal element block. For example:

=====NODAL BASED TETRAHEDRA ELEMENT BLOCK AND MATERIAL INFORMATION =====

Block	Nodal State Index
-----	-----
block_1	1
block_7	2
block_200	3

=====

Block 1, 7, and 200 are using nodal based tetrahedra element formulation. The nodal based stress in block 7 would be found in the nodal state variable `stress_2`. If a node was on a block boundary and attached to elements of block 1 and block 7 then that node would have both `stress_1`, the stress associated with the block 1 material, and `stress_2`, the stress associated with the block 7 material. Note that for nodal based tetrahedra these nodal stresses and material states are the most relevant quantities to visualize. Additionally note that many derived output quantities such as the von Mises stress norm are not hooked to the nodal based output quantities.

6.2.1.2 Q1P0 Formulation

In the Q1P0 element formulation, the internal forces arising from material stress are selectively integrated. Forces arising from the pressure component of the stress are integrated using a single integration point while forces arising from the deviatoric stress are integrated using a 2x2x2 Gauss rule.

The only STRAIN INCREMENTATION option available for this element is STRONGLY_OBJECTIVE.

When post-processing information such as the plastic strain with this element, information at the first integration point should typically be used as it is more accurate than at any other point. The first integration point corresponds to the location at the center of the element where the pressure response is evaluated.



Warning: Material model evaluations at the q1p0 element's deviatoric integration points can result in spurious high and low locked pressures for incompressible (or nearly incompressible) material models. The q1p0 element avoids the pressure locking when calculating the internal forces (for the balance of linear momentum) by discarding the pressures calculated at the deviatoric integration points and replacing them with the pressure from the central integration point. Note that the locked pressures are replaced during element calculations, not inside the constitutive model. This means that material models and element death criteria that fail or accumulate damage based on pressure may be adversely affected by this deviatoric pressure locking. For this reason, the selective-deviatoric (SD) element is generally preferred for material failure analyses. The SD element calculates a single average element volumetric strain and passes that average volumetric strain to all material integration points. The volume averaging of strain in the SD element prevents pressure locking in the material constitutive equations and in the overall element response.

Stress-based values such as `stress` and values derived from it such as `von_mises` are evaluated using a stress tensor taken from a volumetric average of the 8 deviatoric Gauss points for the deviatoric response combined with a pressure response at the central integration point.

The Q1P0 STABILIZATION THRESHOLD command can be used to modify the formulation to



provide additional stabilization as elements become distorted at the cost of accuracy. If a simulation produces inverted elements, these may be able to be mitigated by providing a value of 0.25. One may look at the `stabilization_factor` element variable to determine if this option is being activated in the analysis. A value of 0 in this variable corresponds to a fully q1p0 formulation while a value of 1 corresponds to a fully integrated formulation. Keep in mind that if this is changed from the default value of 0, the formulation is no longer truly q1p0.

Three additional parameters are available to select how the critical time step is evaluated: `Q1P0 TIMESTEP SCALE FACTOR`, `Q1P0 TIMESTEP WAVE SPEED`, and `Q1P0 TIMESTEP LENGTH SCALE`. The critical time step is evaluated using the following formula:

$$\text{timestep} = \text{scale factor} \times \frac{\text{length scale}}{\text{wave speed}}$$

The `Q1P0 TIMESTEP SCALE FACTOR = scale_factor` command scales the calculated time step for elements with this section. The default of 0.95 should be sufficient for almost all analyses. Lowering this slightly may provide better results in certain circumstances. If another time step scale factor is specified within the `PARAMETERS FOR PRESTO REGION` block, they are effectively multiplied together for elements using this section.

The `Q1P0 TIMESTEP WAVE SPEED` command can be used to choose the wave speed used by the time step calculation. The default, `AUTOMATIC`, should be sufficient for all analyses. The `VOLUMETRIC` option calculates wave speed using the bulk modulus while the `SHEAR` option uses the shear modulus. The `AUTOMATIC` option uses the maximum of the other two options.

The `Q1P0 TIMESTEP LENGTH SCALE` command can be used to select the method used to calculate the length scale of the element. The default `MINIMUM_MAPPING_STRETCH` option calculates this as the minimum stretch from the mapping between a unit cube and the current configuration of the element. While this option is relatively slow, it is robust. The `DEFORMED_NODAL_DISTANCE` option calculates this as the minimum non-zero node to node distance within the element. This is the fastest option and a potential increase in speed can be achieved by selecting it at the cost of robustness. The `INSCRIBED_SPHERE_DIAMETER` option calculates this as the diameter of the largest sphere which can fit inside the element.

The `MASS LUMPING METHOD` is used to set the algorithm for determination of the lumped nodal masses. The `ROWSUM` option means first compute the consistent mass matrix then set the lumped mass of each node as the sum of the consistent mass matrix row. The `ROWSUM` method gives more accurate gravity loads and a more consistently stable explicit time step. Row sum is the default method for most elements. The `EVEN` option means take the total element mass and divide it evenly between the elements nodes. The `EVEN` method exists mostly to allow backwards compatibility with previous Sierra/SM versions where the the even mass lumping method was always used. The `DEFAULT` method means the code automatically picks the best mass lumping option based on element type. For most elements this is the row sum method. A few elements such as the under-integrated ten-noded tetrahedron perform better with even mass lumping and use this method as the default.

Note that there also exist other solid element formulations which can be specified in the separate `TOTAL LAGRANGE` section block. These options are currently documented in the in development manual.

6.2.2 Cohesive Section

```
BEGIN COHESIVE SECTION <string>cohesive_section_name
  NUMBER OF INTEGRATION POINTS = <integer>num_int_points(1)
END [COHESIVE SECTION <string>cohesive_section_name]
```



Warning: In order for the cohesive elements to contribute to the global stable time step, a node based time step must be used. See Section 3.2.3 for details.

The `COHESIVE SECTION` command block is used to specify the properties for cohesive zone elements (quadrilateral and triangular). The name of this block (given by `cohesive_section_name`) is referenced by the element block for cohesive elements. If the option for adaptive insertion of cohesive zone elements is used, the name of this block is referenced by the `COHESIVE SECTION` command defined in Section 6.5.5.

`NUMBER OF INTEGRATION POINTS = <integer>num_int_points(1)` specifies the number of integration points for the cohesive element; the default value is one. However, it should be noted that with a single integration point, spurious hourglass-like modes may be introduced to the deformation of the cohesive element. Currently, the quadrilateral cohesive element supports either one or four integration points, while the triangular cohesive element supports either one or three integration points.

6.2.3 Localization Section

```
BEGIN LOCALIZATION SECTION <string>localization_section_name
  MEAN DILATATION FORMULATION
  NUMBER OF INTEGRATION POINTS = <integer>num_int_points
  THICKNESS = <real>thickness
  MEMBRANE FORCES = ON | OFF (ON)
END [LOCALIZATION SECTION <string>localization_section_name]
```



Warning: Currently the localization elements do not contribute to the global stable time step. As such, the user must specify a time step via the `INITIAL TIME STEP` and `TIME STEP INCREASE FACTOR` command lines. See Section 3.1 for details.

6.2.3.1 Kinematics

Localization elements [8] are planar elements that lie between bulk (volumetric) elements and can employ the same underlying bulk material model. Topologically, localization elements are identical to cohesive surface elements. The reason this 2D surface element can access 3D material models is due to the multiplicative decomposition of the deformation gradient $\mathbf{F}$ such that $\mathbf{F} = \mathbf{F}^{\parallel} \mathbf{F}^{\perp}$. Each portion of the decomposition can be expressed as

$$\mathbf{F}^{\perp} = \mathbf{I} + \frac{[[\Phi]]}{h} \otimes \mathbf{N}, \quad \mathbf{F}^{\parallel} = \mathbf{g}_i \otimes \hat{\mathbf{G}}^i \quad (6.1)$$

where $\mathbf{F}^{\parallel}$ encapsulates in-plane stretching and $\mathbf{F}^{\perp}$ reflects the displacement jump $[[\Phi]]$ in the intermediate configuration which can be pushed to the current configuration through $[[\varphi]] = \mathbf{F}^{\parallel} [[\Phi]]$.

Figure 6.2 illustrates the kinematic construction. The jump is normalized by h which one can en-

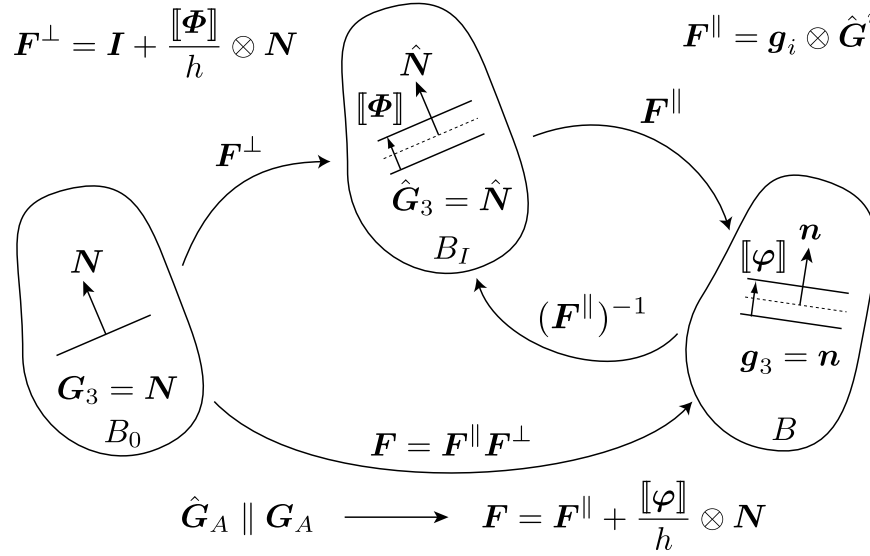


Figure 6.2: The reference B_0 , intermediate B_I , and current configuration B of the body. By re-defining the jump in the intermediate configuration, the formulation simplifies to an additive decomposition having contributions from membrane stretching and the jump.

vision as an element thickness or a characteristic length scale governing separation. Quantities are considered to be constant through the thickness h . The curvilinear basis vectors in the reference and current configuration are G_A and g_i , respectively. Given that N is constructed to be normal to the in-plane basis vectors G_A in the reference configuration, one can prove that the in-plane basis vectors in the intermediate configuration are equivalent to the in-plane basis vectors in the reference configuration ($\hat{G}_A = F^\perp G_A = G_A$). One can then express the multiplicative decomposition as an additive decomposition thus simplifying the initial formulation given in [8].

$$F = F^\parallel + \frac{[[\varphi]]}{h} \otimes N \quad (6.2)$$

Because the length scale h is independent of the discretization, the methodology is regularized and ideal for employing a local, softening material model to simulate the failure process. Details concerning the force calculation are available in [8].

6.2.3.2 Command summary

The `LOCALIZATION SECTION` command block is used to specify the properties for localization elements (quadrilateral and triangular). The name of this block (given by `localization_section_name`) is referenced by the element block for localization elements.

MEAN DILATATION FORMULATION This command line will yield a constant pressure formulation. The average pressure is obtained at all integration points through a modification of the kinematic quantities. For hypo-elastic materials, $\text{tr}[D]$ is volume-averaged. For uncoupled hyperelastic models, $\det[F]$ is volume-averaged. After volume averaging $\text{tr}[D]$ and $\det[F]$, local

dilatational contributions are removed, and additively (hypoelastic) or multiplicatively (hyperelastic) include the volume-averaged response. This results in an average pressure (by construction) and has been shown to be effective in avoiding element locking during isochoric deformations. Note that one can use a single integration point to achieve a constant pressure, but hourglass control is not provided to suppress spurious modes. If isochoric deformations are of concern, it is recommended to use a fully-integrated element with the `MEAN DILATATION FORMULATION`. The `MEAN DILATATION FORMULATION` will only be applied if the command is given in the `LOCALIZATION SECTION`.

`NUMBER OF INTEGRATION POINTS = <integer>num_int_points` The default number of integration points for a localization element depends on the topology, but is always sufficient for full integration. For an eight-noded hexahedron (planar four-noded quadrilateral) the default is four integration points while for a six-noded wedge (planar three-noded triangle) the default is one integration point. However, it should be noted that with a single integration point, spurious hourglass modes can be introduced into the deformation of the quadrilateral localization element. Currently, the quadrilateral localization element supports one and four integration points while the triangular localization element supports one integration point.

`THICKNESS = <real>thickness` The `THICKNESS` command line sets the localization element thickness h . In many respects, h should be considered a material parameter as it governs the evolution of surface separation. The introduction of h generates the true length scale in the problem: the process zone size. Care must be taken to adequately resolve the process zone size or the methodology will not be regularized.

`MEMBRANE FORCES = ON | OFF (ON)` The `MEMBRANE FORCES` command line should be used when the element size l is on the order of h . In this specific case, the membrane forces will be sufficiently large to corrupt the crack-tip fields. If $h \sim l$, we recommend that the membrane forces be `OFF`. Turning off membrane forces alters the force calculation to only employ the referential traction $\mathbf{T} = \mathbf{P}\mathbf{N}$. In this sense, the formulation becomes “cohesive”. But because $\mathbf{F}^{\parallel}$ is still employed in the kinematics to capture critical quantities such as the triaxiality, the stiffness matrix is no longer symmetric; however, the residual is correct. Numerous studies in implicit quasi-statics and implicit dynamics have confirmed that CG is still robust. To avoid non-monotonic reductions in the residual, one might employ solvers that can handle non-symmetric systems (ML) and solution methods other than CG (Newton with a line search) to obtain improved rates of convergence.



Warning: Domain decomposition may generate domains in which localization elements are isolated. Neither the upper or lower surface is connected to a bulk element. Because the localization element does contain a zero-energy membrane mode that is suppressed through bulk element assembly, the user will receive warnings that indicate additional rigid body modes on particular processors. To remedy the issue, the user can employ tangent diagonal scaling in the preconditioner. Studies have shown this solution to be effective with minimal impact on the rate of convergence.

6.2.3.3 Usage guidelines

In contrast to cohesive surface elements, localization elements can employ bulk constitutive models for surface separation. For a typical application, the analyst might

1. Identify a constitutive model that captures the failure process. This might include a local damage model or any model that employs strain softening to facilitate strain localization.
2. Conduct mesh-dependent simulations with bulk elements of size s to understand potential paths for crack initiation and growth in specimen geometries targeted for parameterizing constitutive model parameters.
3. Seed potential paths in specimen geometries and incorporate h into the fitting process. The fitting process may not be unique in that the same far-field response might be obtained from multiple combinations of both h and the material parameters that govern the failure process. An initial guess to being the iterative process might include $h = 1/4s$. Ensure that for all cases of h that the process zone that evolves is resolved.
4. Consider the possibility of only enabling strain softening in the constitutive model that drives surface separation. Although this will assure that localization will occur in the surface elements, regulating softening to the seeded plane can introduce non-smoothness in the failure process.
5. Understand the impact of h . If h is too large, the fields governing the failure process will be less smooth and the additional compliance can create additional nonlinearities that can complicate implicit solution. Please consider refitting model parameters with lower values of h . If h is too small, the localization element thickness will drive the condition number of the implicit system. item Explore component or system level geometries with seeded paths for crack initiation and propagation. Additional simulations with bulk elements may be conducted to highlight potential failure paths. Refine the mesh to ensure that the far-field predictions are indeed mesh independent and that the process zone that evolves from the given micromechanics regularized by the length scale h is resolved.
6. Inspect of the process zone with a focus on the local metrics that drive the failure process. For example, if stress triaxiality drives void growth, one should examine fields of triaxiality in both the bulk and localization elements. In contrast to other methods like the J -integral that focus on a driving force evaluated at some radius from the crack tip, localization elements seek to capture the resistance at the crack tip. Consequently, crack-tip fields must be resolved.
7. Reflect on the fields employed for model parameterization and the fields evolving in component and system level models. If possible, align field evolution in component/system geometries with field evolution in specimen geometries. Disparities may drive the need for additional calibration experiments.

Although these usage guidelines have not focused on incorporating stochastic processes, one may sample distributions in material parameters. The inclusion of a method for regularization enables

such findings in that the mesh-dependence associated with fracture/failure is not convoluted with a stochastic representation of the micro-mechanical process.

6.2.4 Shell Section

```
BEGIN SHELL SECTION <string>shell_section_name
  THICKNESS = <real>shell_thickness
  THICKNESS MESH VARIABLE =
    <string>THICKNESS|<string>var_name
  THICKNESS TIME STEP = <real>time_value
  THICKNESS SCALE FACTOR = <real>thick_scale_factor(1.0)
  INTEGRATION RULE = TRAPEZOID|GAUSS|LOBATTO|SIMPSONS|
    USER|ANALYTIC|CODE_SELECTED(CODE_SELECTED)
  NUMBER OF INTEGRATION POINTS = <integer>num_int_points(5)
  FORMULATION = KH_SHELL|BT_SHELL|BL_SHELL|NQUAD|BT_SC_SHELL
    THICKSHELL|SEVEN_PARAMETER|FIBER_SHELL|C0_TRI_SHELL|
    ORIG_TRI_SHELL (BT_SHELL``QUAD`` or C0_TRI_SHELL``Tri``)
  BEGIN USER INTEGRATION RULE
    <real>location_1 <real>weight_1
    <real>location_2 <real>weight_2
    .
    .
    <real>location_n <real>weight_n
  END [USER INTEGRATION RULE]
  LOFTING FACTOR = <real>lofting_factor(0.5)
  OFFSET MESH VARIABLE = <string>var_name
  ORIENTATION = <string>orientation_name
  DRILLING STIFFNESS FACTOR = <real>stiffness_factor(1.0e-5)
  RIGID BODY = <string>rigid_body_name
  RIGID BODIES FROM ATTRIBUTES = <integer>first_id
    TO <integer>last_id
  STABLE TIME STEP ESTIMATE = COURANT|GERSCHGORIN(GERSCHGORIN)

  CONTACT THICKNESS = <real>contactThickness
  CONTACT LOFTING FACTOR = <real>contactLofting

  FIBER ORIENTATION = <real>angle(0)
  R VECTOR READ VARIABLE = <string>rVecName
  T VECTOR READ VARIABLE = <string>tVecName

END [SHELL SECTION <string>shell_section_name]
```

👉 **Explicit**

The `SHELL SECTION` command block is used to specify the properties for a shell element. If this command block is referenced in an element block of three-dimensional, four-node elements, the elements in the block will be treated as shell elements. The parameter, `shell_section_name`, is user-defined and is referenced by a `SECTION` command line in a `PARAMETERS FOR BLOCK` command block.

Either a `THICKNESS` command line or a `THICKNESS MESH VARIABLE` command line must appear in the `SHELL SECTION` command block.

If a shell element block references a `SHELL SECTION` command block with the command line:

```
THICKNESS = <real>shell_thickness
```

then the thickness of all shell elements in the block will be initialized to the value `shell_thickness`.

Sierra/SM can also initialize the thickness using an attribute defined on elements in the mesh file. Meshing programs such as PATRAN and Cubit typically set the element thickness as an attribute on the elements. If the elements have one and only one attribute defined on the mesh, the `THICKNESS MESH VARIABLE` command line should be specified as

```
THICKNESS MESH VARIABLE = THICKNESS
```

which causes the thickness of the element to be initialized to the value of the attribute for that element. If there are zero attributes or more than one attribute, the thickness variable will not be automatically defined, and the command will fail.

The thickness may also be initialized by any other field present on the input mesh. To specify a field other than the single-element attribute, use the following form of the `THICKNESS MESH VARIABLE` command line:

```
THICKNESS MESH VARIABLE = <string>var_name
```

Here, the string `var_name` is the name of the initializing field.

The input mesh may have values defined at more than one point in time. To choose the point in time in the mesh file that the variable should be read, use the command line

```
THICKNESS TIME STEP = <real>time_value
```

The default time point in the mesh file at which the variable is read is 0.0.

Once the thickness of a shell element is initialized by using either the `THICKNESS` command line or the `THICKNESS MESH VARIABLE` command line, this initial thickness value can then be scaled using the scale-factor command line:

```
THICKNESS SCALE FACTOR = <real>thick_scale_factor
```

If the initial thickness of the shell is 0.15 inch, and the value for `thick_scale_factor` is 0.5, then the scaled thickness of the membrane will be 0.075.

The thickness mesh variable specification may be coupled with the `THICKNESS SCALE FACTOR` command line. In this case, the thickness mesh variable is scaled by the specified factor.

The shell formulation can be selected via the `FORMULATION` command line. Several options are available:

- The `BT_SHELL` option is used to select the Belytschko-Tsay (BT) shell formulation. This is the default. The BT shell formulation has constant transverse shear, and is the fastest shell element available. Note, this element can experience stability issues at large in-plane strains.

- The `BL_SHELL` option is used to select the Belytschko-Leviathan (BL) shell formulation. The BL shell element is similar to the Belytschko-Tsay element but has additional shear terms and hourglass controls that the BT does not have. It also includes a projection of the angular velocities and the internal forces. The BL can better solve problems with high gradients of transverse shear (such as twisted beams), and no hourglass control or drilling stiffness is needed. Note, this element can experience stability issues at large in-plane strains.
- The `KH_SHELL` option specifies the Key-Hoff shell formulation, which includes a nonlinear transverse shear term. The Key-Hoff shell can better solve problems with high gradients of transverse shear (such as twisted beams), but distorted (non-rectangular) elements can be excessively stiff. Note, this element can experience stability issues at large in-plane strains.
- The `BT_SC_SHELL` option selects the Belytschko-Tsay formulation and adds an additional strongly co-rotational formulation for in plane stretching and shear. This option can remedy the problems seen with large in-plane strains in the standard BL, KH, and BT shell formulations. However, this option adds significant cost. Additionally, this option has not been thoroughly verified; it should be considered “in-development” and used with caution.
- The `NQUAD` option selects a formulation for a completely elastic plate.
- The `THICKSHELL` option selects an experimental shell element that treats hexahedral elements with a shell-based formulation, the option is in development and generally should not be used in production runs.
- The `SEVEN_PARAMETER` option selects an experimental shell formulation, the option is in development and generally should not be used in production runs.
- The `FIBER_SHELL` option is an experimental fiber reinforced laminate shell. The option is in development and generally should not be used in production runs. The option only functions with a handful of specialized materials.
- The `CO_TRI_SHELL` option is used to specify the triangular shell formulation developed by Kennedy, Belytschko, and Lin (1986). The C^0 tri shell is a robust, versatile element that uses an incremental strain computation, making it applicable to large strains. Note that this element has no resistance to rotations about the axis perpendicular to its plane.
- The `TRI_SHELL` option is a specialized triangular shell formulation. The option is in development and generally should not be used in production runs.

For shell elements, the user may select from a number of integration rules, including a user-defined integration option. The integration rule is selected with the command line

```
INTEGRATION RULE = <string>TRAPEZOID|GAUSS|LOBATTO|SIMPSONS|
ANALYTIC|CODE_SELECTED|USER(CODE_SELECTED)
```

Consult the element documentation [5] for a description of different integration schemes for shell elements.

For most material models the code will by default select the five point `TRAPEZOID` through the thickness integration rule. The code selected integration scheme for the elastic material model is `ANALYTIC`, which is an analytic through-thickness integration scheme. This scheme is significantly faster and slightly more accurate than any of the point-wise rules but only valid for fully elastic shell formulations.

Shells using the analytic integration rule do not compute integration point material or stress quantities such as stress, strain, or the various quantities derived from stress and strain. If integration

point quantities are desired for elastic material shells, the shells can be forced to use a point-wise integration scheme by manually setting the integration rule to `TRAPEZOID` or another point-wise rule.

Currently the analytic integration scheme is only available for the elastic material model when using the `BT_SHELL` shell formulation for quad shell elements.

The number of integration points for the piecewise integration point rules can be set to any number greater than one by using the following command line:

```
NUMBER OF INTEGRATION POINTS = <integer>num_int_points(5)
```

The `SIMPSONS`, `GAUSS`, and `LOBATTO` integration schemes in the `INTEGRATION RULE` command line all default to five integration points. The number of integration points for these three schemes can be reset by using the `NUMBER OF INTEGRATION POINTS` command line. There are limitations on the number of integration points for some of these integration rules. The `SIMPSONS` rule can be set to any number greater than one, the `GAUSS` scheme can be set to one through seven integration points, and the `LOBATTO` integration scheme can be set to two through seven integration points.

In addition to these standard integration schemes, the `USER INTEGRATION RULE` command block may be used to define a custom integration scheme.

```
BEGIN USER INTEGRATION RULE
  <real>location_1 <real>weight_1
  <real>location_2 <real>weight_2
  .
  .
  <real>location_n <real>weight_n
END [USER INTEGRATION RULE]
```

A standard integration scheme and a user scheme may *not* be specified simultaneously. If the `USER` option is specified in the `INTEGRATION RULE` command line, a set of integration locations with associated weight factors must be specified. This is done with tabular input command lines inside the `USER INTEGRATION RULE` command block. The number of command lines inside this command block should match the number of integration points specified in the `NUMBER OF INTEGRATION POINTS` command line. For example, to use a user-defined scheme with three integration points, the `NUMBER OF INTEGRATION POINTS` command line should specify three (3) integration points and the number of command lines inside the `USER INTEGRATION RULE` command block should be three (to give three locations and three weight factors).

For the user-defined rule, the integration point locations should fall between -1.0 and 1.0 , and the weights should sum to 1.0 .

The command line

```
LOFTING FACTOR = <real>lofting_factor(0.5)
```

allows the user to shift the location of the mid-surface of a shell element relative to the geometric location of the shell element. By default, the geometric location of a shell element in a mesh

represents the mid-surface of the shell. If a shell has a thickness of 0.2 inch, the top surface of the shell is 0.1 inch above the geometric surface defined by the shell element, and the bottom surface of the shell is 0.1 inch below the geometric surface defined by the shell element. (The top surface of the shell is the surface with a positive element normal; the bottom surface of the shell is the surface with a negative element normal.)

Figure 6.3 shows an edge-on view of shell elements with a thickness of t and the location of the geometric plane in relation to the shell surfaces for three different values of the lofting factor—0.0, 0.5, and 1.0. For a lofting factor of 0.0, the geometric surface defined by the shell corresponds to the top surface of the shell element. A lofting factor of 1.0 puts the geometric surface at the bottom surface of the shell element. The geometric surface is midway between the top and bottom surfaces for a lofting factor of 0.5, which is the default. Lofting factors greater than 1.0 or less than 0.0 are typically reserved for layering of shells—otherwise, this is not recommended.

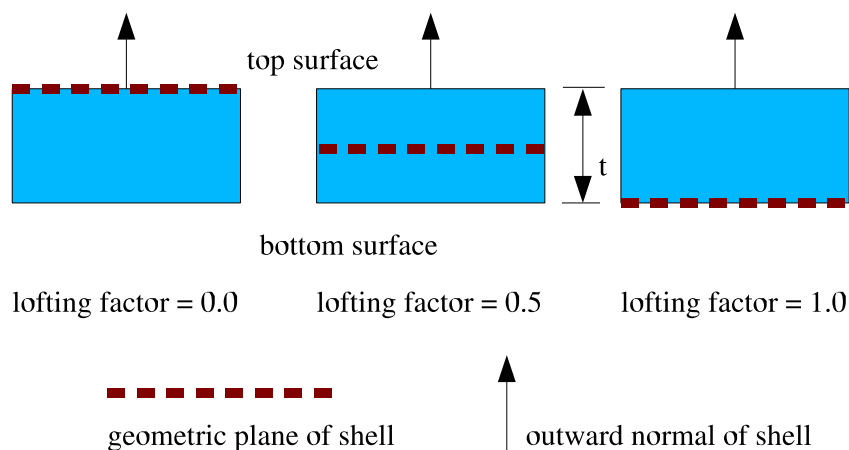


Figure 6.3: Location of geometric plane of shell for various lofting factors.

Consider the example of a lofting factor set to 1.0 for a shell with thickness of 0.2 inch. In this case, the top surface of the shell will be located at a distance of 0.2 inch from the geometric surface (measuring in the direction of the positive shell normal), and the bottom surface will be located at the geometric surface.

Both the shell mechanics and contact use shell lofting. See Section 8.4 for a discussion of lofting surfaces for shells and contact. It is recommended that shell lofting values other than 0.5 not be used if the shell is thick. If the shell is thicker than its in-plane width, the shell lofting algorithms may become unstable.

Lofting as described above may also be implemented through the command

```
OFFSET MESH VARIABLE = <string>var_name
```

which allows an offset value to be read from an attribute (or variable) on the mesh file. This attribute (or variable) must be named but may have any name, and this name is specified in place of `var_name` in the command. An offset is a dimensional (i.e., not scaled by the shell thickness) value that gives the shell mid-plane shift in the positive shell normal direction. Internal to Sierra/SM

the offset value is converted to an equivalent lofting factor by dividing by the initial thickness and adding 0.5. Thus, an offset of zero gives a lofting factor of 0.5, an offset of one half of the thickness gives a lofting factor of one, and an offset of negative one half of the thickness gives a lofting factor of zero. There is no check that given offset values produce lofting values between zero and one, which allows any offset to be specified but requires that care be exercised to avoid unstable lofted shells.



Warning: An offset and a lofting factor may not both be specified in the same shell section. Both determine the shell lofting and may conflict.

The `ORIENTATION` command line lets the user select a coordinate system for output of in-plane stresses and strains. The `ORIENTATION` option makes use of an embedded coordinate system r, s, t associated with each shell element. The r, s, t coordinate system for a shell element is shown in Figure 6.4. The r -axis extends from the center of the shell to the midpoint of the side of the shell defined by nodes 1 and 2. The t -axis is located at the center of the shell and is normal to the surface of the shell at the center point. The s -axis is the cross-product of the t -axis and the r -axis. The r, s, t -axes form a local coordinate system at the center of the shell; this local coordinate system moves with the shell element as the element deforms.

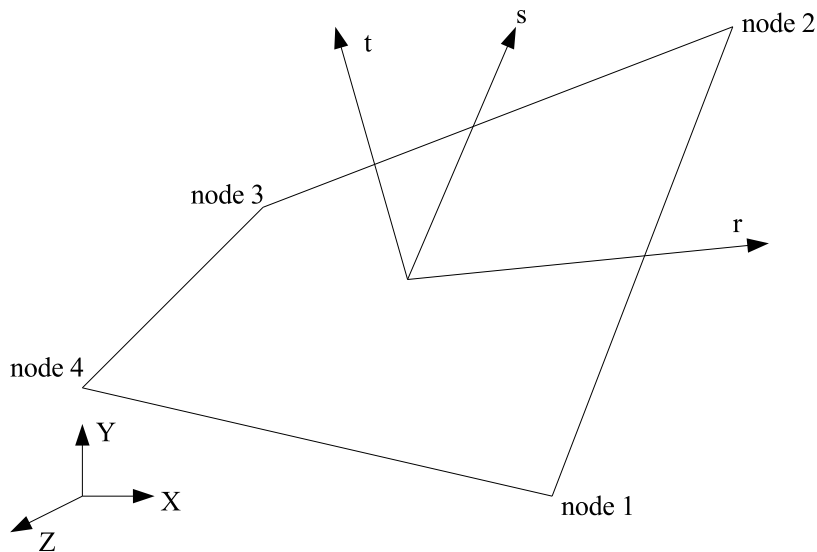


Figure 6.4: Local rst coordinate system for a shell element.

The `ORIENTATION` command line in the `SHELL SECTION` command block references an `ORIENTATION` command block that appears in the `SIERRA` scope. As described in Chapter 2 of this document, the `ORIENTATION` command block can be used to define a local co-rotational coordinate system $X''Y''Z''$ at the center of a shell element. In the original shell configuration (time 0), one of the axes— X'' , Y'' , or Z'' —is projected onto the plane of the shell element. The angle between this projected axis of the $X''Y''Z''$ coordinate system and the r -axis is used to establish the transformation for in-plane stresses and strains. We will illustrate this with an example.

Suppose that in an `ORIENTATION` command block a rotation of 30 degrees about the 1-axis (X' -axis) has been specified. The command line for this rotation in the `ORIENTATION` command block would be

```
ROTATION ABOUT 1 = 30
```

For this case, the Y'' -axis is projected onto the plane of the shell (Figure 6.5). The angle between this projection and the r -axis establishes a transformation for the in-plane stresses of the shell (the stresses in the center of the shell lying in the plane of the shell). What will be output as the in-plane stress σ_{xx}^{ip} will be in the Y'' -direction; what will be output as the in-plane stress σ_{yy}^{ip} will be in the Z'' -direction. The in-plane stress σ_{xy}^{ip} is a shear stress in the $Y''Z''$ -plane. The $X''Y''Z''$ coordinate system maintains the same relative position in regard to the rst coordinate system. This means that the $X''Y''Z''$ coordinate system is a local coordinate system that moves with the shell element as the element deforms.

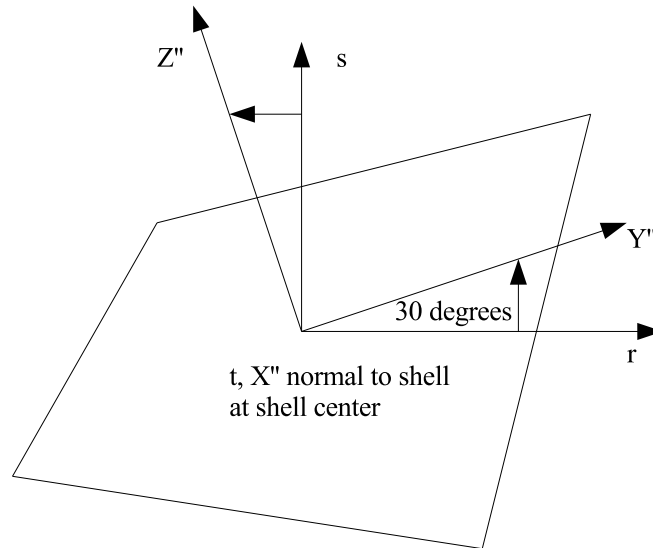


Figure 6.5: Rotation of 30 degrees about the 1-axis (X' -axis).

The following permutations for output of the in-plane stresses occur depending on the axis (1, 2, or 3) specified in the `ROTATION ABOUT` command line:

- Rotation about the 1-axis (X' -axis): The in-plane stress σ_{xx}^{ip} will be in the Y'' -direction; the in-plane stress σ_{yy}^{ip} will be in the Z'' -direction. The in-plane stress σ_{xy}^{ip} is a shear stress in the $Y''Z''$ -plane.
- Rotation about the 2-axis (Y' -axis): The in-plane stress σ_{xx}^{ip} will be in the Z'' -direction; the in-plane stress σ_{yy}^{ip} will be in the X'' -direction. The in-plane stress σ_{xy}^{ip} is a shear stress in the $Z''X''$ -plane.
- Rotation about the 3-axis (Z' -axis): The in-plane stress σ_{xx}^{ip} will be in the X'' -direction; the in-plane stress σ_{yy}^{ip} will be in the Y'' -direction. The in-plane stress σ_{xy}^{ip} is a shear stress in the $X''Y''$ -plane.

If no orientation is specified, the in-plane stresses and strains are output in the consistent axes from a default orientation, which is a rectangular system with the X' and Y' axes taken as the global X and Y axes, respectively, and `ROTATION ABOUT 1 = 0.0`.

The command line

```
DRILLING STIFFNESS FACTOR = <real>stiffness_factor
```

adds stiffness in the drilling degrees of freedom to quadrilateral shells. Drilling degrees of freedom are rotational degrees of freedom in the direction orthogonal to the plane of the shell at each node. The formulation used for the quadrilateral shells has no rotational stiffness in this direction. This can lead to spurious zero-energy modes of deformation similar in nature to hourglass deformation. This makes obtaining a solution difficult in quasistatic problems and can result in singularities when using the full tangent preconditioner.

The `stiffness_factor` should be chosen as a quantity small enough to add enough stiffness to allow the solve to be successful without unduly affecting the solution. The default value for `stiffness_factor` is `1.0e-5`, which places drilling stiffness about 10000X lower than the bending stiffness. If singularities are encountered in the solution or hourglass-like deformation is observed in the drilling degrees of freedom, setting a larger value may be appropriate.

Elements in a section can be made rigid by including the `RIGID BODY` command line. The `RIGID BODY` command line specifies an indenter that maps to a rigid body command block. See Section 6.3.1 for a full discussion of how to create rigid bodies and Section 6.3.1.1 for information on the use of the `RIGID BODIES FROM ATTRIBUTES` command.

The command line `STABLE TIME STEP ESTIMATE` specifies the method used to calculate the size of the critical time step. The default method is `GERSCHGORIN` circle theorem, which is used to find a bound to the maximum membrane, shear and bending frequencies of the problem. The `COURANT` method uses the characteristic length of the element and the membrane wave speed to approximate the stable time step. This estimate tends to be less conservative than `GERSCHGORIN`, but might not be appropriate for problems with dominant shear or bending responses. This command line option is only available for quadrilateral shells. Triangular shells will only use the Courant estimate.

The command lines `CONTACT SHELL THICKNESS` and `CONTACT SHELL LOFTING` can be used to set the thickness and lofting factor for use in contact. This can be a different thickness or lofting factor than used for defining the element stiffness.

6.2.4.1 Orthotropic Properties

Orthotropic shell properties have an effect on orthotropic shell materials such as the orthotropic elastic shell material 5.2.26. The orthotropic shell properties may be defined by using the `FIBER ORIENTATION`, `R VECTOR READ VARIABLE`, and `T VECTOR READ VARIABLE` commands. The `R` vector and `T` vectors of this coordinate system are defined individually on each element in the input mesh element fields with names given by `R VECTOR READ VARIABLE` and `T VECTOR READ VARIABLE`.

The input `R` and `T` vectors are turned into the element local `RST` coordinate system via the process shown in Figure 6.6. The construction of the initial `RST` coordinate system is done in undeformed coordinates. As the shell moves during the course of the analysis the local `RST` coordinate system



will move with it. First the true T vector is determined for the coordinate system. The T vector must lie along either the positive or negative shell normal. The input T vector decides only if the up or down shell normal is selected as T' . Next the input R vector is restricted to the plane of the shell. This operation makes R' the closest vector to the input R that is orthogonal to T' . The S' vector is constructed by $T' \times R'$. The S' is by construction orthogonal to both R' and T' . Finally the R' and S' vectors are rotated about the T' axis by the `FIBER ORIENTATION` angle Θ to construct the final local RST coordinate system. This procedure ensures that the model has a properly defined RST coordinate system in the undeformed configuration and can be visualized by outputting and plotting on the element '`r_vector`', '`s_vector`' and '`t_vector`' fields.

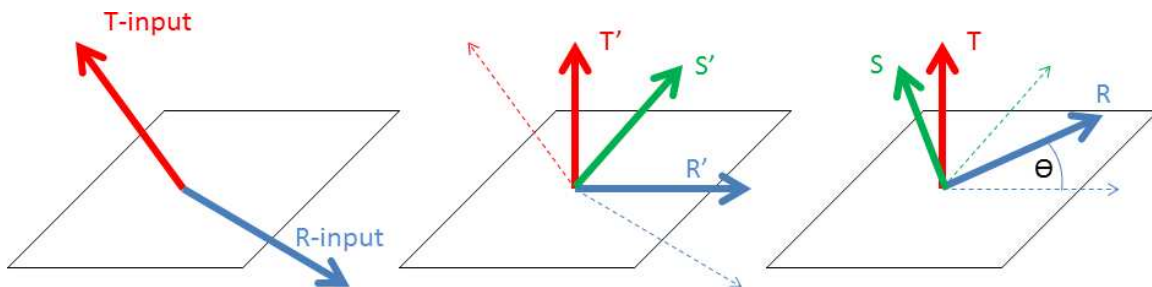


Figure 6.6: Definition of shell RST coordinate system

6.2.5 Layered Shell Section

```
BEGIN LAYERED SHELL SECTION <string>sect
  LAYER <int>layerId = <string>subSectName
                        <string>material <string>model
  LOFTING FACTOR = <real>loft(0.5)
  CONTACT THICKNESS = <real>contactThickness
  CONTACT LOFTING FACTOR = <real>contactLofting
END
```

The `LAYERED SHELL SECTION` command block is used to define a single shell element that is a composite of layers with different properties. The layers of the shell can have different materials and different shell formulations.

Internally to the code a layered shell is represented by several individual elements that share the same nodes and same topology. Layered shells will create new element blocks and new elements in the output file to represent each distinct shell layer. Boundary conditions such as contact or element death can be performed on either the original shell element or on the individual layers specifying the appropriate block name.

The `LAYER` defines a layer for the current shell. The layers of the shell will be stacked from the bottom to the top based on the order of the `LAYER` commands on the input deck. A shell may have up to 99 different layers defined. The `layerId` input is a positive integer provided by the user. This number will be used to label the shell layers to aid in visualization. The new element block sub-layers will be named based on the provided `layerId` using the convention

```
<block_name>_SIERRA_CREATED_LAYER<layerID>
```


The `subSectName` is the name of the shell section associated with the layer. The shell section will be used to define the thickness of the layer and any shell formulation options to use in the layer. The `material` and `model` names define the material and material model that the layer is composed of. Any material that is valid for shell elements can be used as a shell layer material. A default `material` and `model` may be provided in the `parameters` for `block` command block. Any layer in the section that does not define a material and model will use this default material and model.

Note that the first order approximations used in the shell element formulation does not accurately compute transverse shear strains for layered shells. This is handled and corrected internally by applying a shear correction factor [9].

The lofting factor for the entire shell stack can be specified with the `LOFTING FACTOR` option. Similar to standard shells this lofting factor can vary from 0.0 to 1.0 and has a default value of 0.5 (shell element passes through mid-plane of the layered shell stack.) The lofting of the individual layers will be computed automatically based on the layer thicknesses and layered shell lofting factor.

The command lines `CONTACT SHELL THICKNESS` and `CONTACT SHELL LOFTING` can be used to set the thickness and lofting factor for use in contact. These commands define the contact behavior of the layered shell stack as a whole.



Warning: The ability to use element death on a layered shell block has not yet been implemented. If `DEATH ON INVERSION` is active, the problem may still fail if the layered shells invert.

Example of layered shell syntax:

```
begin shell section SHELL_1
  thickness = 0.1
end shell section SHELL_1

begin shell section SHELL_2
  thickness = 0.2
end shell section SHELL_2

begin shell section SHELL_3
  thickness = 0.1
end shell section SHELL_3

begin layered shell section LAYERED_SHELL
  layer 1 = SHELL_1 steel elastic
  layer 2 = SHELL_2 copper elastic_plastic
  lofting factor = 1.0
end
```

```
begin parameters for block block 1
```

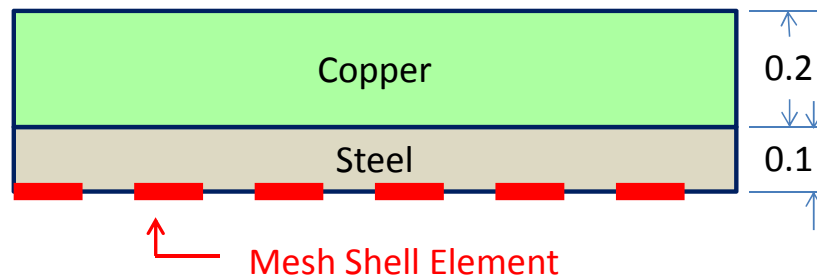


Figure 6.7: Layered Shell Example

6.2.6 Membrane Section

```
BEGIN MEMBRANE SECTION <string>membrane_section_name
  THICKNESS = <real>mem_thickness
  THICKNESS MESH VARIABLE =
    <string>THICKNESS|<string>var_name
  THICKNESS TIME STEP = <real>time_value
  THICKNESS SCALE FACTOR = <real>thick_scale_factor(1.0)
  FORMULATION = <string>MEAN_QUADRATURE|
    SELECTIVE_DEVIATORIC(MEAN_QUADRATURE)
  DEVIATORIC PARAMETER = <real>deviatoric_param
  LOFTING FACTOR = <real>lofting_factor(0.5)
  RIGID BODY = <string>rigid_body_name
  RIGID BODIES FROM ATTRIBUTES = <integer>first_id
    TO <integer>last_id
  CONTACT THICKNESS = <real>contactThickness
  CONTACT LOFTING FACTOR = <real>contactLofting
END [MEMBRANE SECTION <string>membrane_section_name]
```

The `MEMBRANE SECTION` command block is used to specify the properties for a membrane element. If a section defined by this command block is referenced in the parameters for a block of four-noded elements, the elements in that block will be treated as membranes. The parameter `membrane_section_name` is user-defined and is referenced by a `SECTION` command line in a `PARAMETERS FOR BLOCK` command block.

Either a `THICKNESS` command line or a `THICKNESS MESH VARIABLE` command line must appear in the `MEMBRANE SECTION` command block.

If a membrane element block references a `MEMBRANE SECTION` command block with the command line

```
THICKNESS = <real>mem_thickness
```

then all the membrane elements in the block will have their thickness initialized to the value `mem_thickness`.

Sierra/SM can also initialize the thickness using an attribute defined on elements in the mesh file. Meshing programs such as PATRAN and CUBIT typically set the element thickness as an attribute on the elements. If the elements have one and only one attribute defined on the mesh, the `THICKNESS MESH VARIABLE` command line should be specified as

```
THICKNESS MESH VARIABLE = THICKNESS
```

which causes the thickness of the element to be initialized to the value of the attribute for that element. If there are zero attributes or more than one attribute, the thickness variable will not be automatically defined, and the command will fail.

The thickness may also be initialized by any other field present on the input mesh. To specify a field other than the single-element attribute, use this form of the `THICKNESS MESH VARIABLE` command line:

```
THICKNESS MESH VARIABLE = <string>var_name
```

where the string `var_name` is the name of the initializing field.

The input mesh may have values defined at more than one point in time. To choose the point in time in the mesh file that the variable should be read, use the command line:

```
THICKNESS TIME STEP = <real>time_value
```

The default time point in the mesh file at which the variable is read is 0.0.

Once the thickness of a membrane element is initialized by using either the `THICKNESS` command line or the `THICKNESS MESH VARIABLE` command line, this initial thickness value can then be scaled by using the scale factor command line:

```
THICKNESS SCALE FACTOR = <real>thick_scale_factor
```

If the initial thickness of the membrane is 0.15 units, and the value for `thick_scale_factor` is 0.5, then the scaled thickness of the membrane will be 0.075 units.

The `FORMULATION` command line specifies whether the element will use a single-point integration rule (mean quadrature) or a selective-deviatoric integration rule:

```
FORMULATION = <string>MEAN_QUADRATURE|SELECTIVE_DEVIATORIC  
(MEAN_QUADRATURE)
```

If the user wishes to use the selective-deviatoric rule, the `DEVIATORIC PARAMETER` command line must also appear in the `MEMBRANE SECTION` command block:

```
DEVIATORIC PARAMETER = <real>deviatoric_param
```

The selective-deviatoric elements, when used with a parameter greater than 0.0, provide hourglass control without artificial hourglass parameters. The selective-deviatoric parameter, `deviatoric_param`, which is valid from 0.0 to 1.0, indicates how much of the deviatoric response should be taken from a uniform-gradient integration and how much should be taken from a full integration of the element. A value of 0.0 will give a pure uniform-gradient response with no hourglass control. Thus, this value is of little practical use. A value of 1.0 will give a fully integrated deviatoric response. Although any value between 0.0 and 1.0 is perfectly valid, lower values are generally preferred.

The command line

```
LOFTING FACTOR = <real>lofting_factor(0.5)
```

allows the user to shift the location of the mid-surface of a membrane element relative to the geometric location of the membrane element. By default, the geometric location of a membrane element in a mesh represents the mid-surface of the membrane. If a membrane has a thickness of 0.2 inch, the top surface of the membrane is 0.1 inch above the geometric surface defined by the membrane element, and the bottom surface of the membrane is 0.1 inch below the geometric surface defined by the membrane element. (The top surface of the membrane is the surface with a positive element normal; the bottom surface of the membrane is the surface with a negative element normal.)

Figure 6.3 illustrates shell lofting, which is also applicable to membranes. For membranes, Figure 6.3 represents an edge-on view of membrane elements with a thickness of t and the location of the geometric plane in relation to the membrane surfaces for three different values of the lofting factor—0.0, 0.5, and 1.0. For a lofting factor of 0.0, the geometric surface defined by the membrane corresponds to the top surface of the membrane element. A lofting factor of 1.0 puts the geometric surface at the bottom surface of the membrane element. The geometric surface is midway between the top and bottom surfaces for a lofting factor of 0.5, which is the default.

Consider the example of a lofting factor set to 1.0 for a membrane with thickness of 0.2 inch. In this case, the top surface of the membrane will be located at a distance of 0.2 inch from the geometric surface (measuring in the direction of the positive shell normal), and the bottom surface will be located at the geometric surface.

Both the membrane mechanics and contact use membrane lofting. See Section 8.4 for a discussion of lofting surfaces for membranes and contact.

Elements in a section can be made rigid by including the `RIGID BODY` command line. The `RIGID BODY` command line specifies an indenter that maps to a rigid body command block. Consult Section 6.3.1 for a full description of how to create rigid bodies and Section 6.3.1.1 for information on the use of the `RIGID BODIES FROM ATTRIBUTES` command.

The command lines `CONTACT SHELL THICKNESS` and `CONTACT SHELL LOFTING` can be used to set the thickness and lofting factor for use in contact. The thickness used for contact can be different than the thickness used for the element stiffness calculations.

6.2.7 Beam Section

```
BEGIN BEAM SECTION <string>beam_section_name
    SECTION = ROD|TUBE|BAR|BOX|I|L|T|T1|C1|C2|Z|HAT|I2
#
# Section dimensions, definition depends on particular section type.
#
    D1 = <real>d1
    D1 VARIABLE = <STRING>d1_var
    D2 = <real>d2
    D2 VARIABLE = <STRING>d2_var
    D3 = <real>d3
    D3 VARIABLE = <STRING>d3_var
    D4 = <real>d4
    D4 VARIABLE = <STRING>d4_var
    D5 = <real>d5
    D5 VARIABLE = <STRING>d5_var
    D6 = <real>d6
    D6 VARIABLE = <STRING>d6_var

    T AXIS = <real>tx <real>ty <real>tz
    T AXIS VARIABLE = <string>t_axis_var
    REFERENCE AXIS = <string>CENTER|RIGHT|
        TOP|LEFT|BOTTOM(CENTER)
    AXIS OFFSET = <real>s_offset <real>t_offset
    AXIS OFFSET GLOBAL = <real>x_offset <real>y_offset <real>z_offset
    AXIS OFFSET VARIABLE = <string>axis_offset_var
    RIGID BODY = <string>rigid_body_name
    RIGID BODIES FROM ATTRIBUTES = <integer>first_id
        TO <integer>last_id
    VISUALIZE INTEGRATION POINTS = OFF|ON(OFF)
     Explicit ROTATIONAL MASS SCALING = OFF|ON(ON)
END [BEAM SECTION <string>beam_section_name]
```

The `BEAM SECTION` command block is used to specify the properties for a beam element. If this command block is referenced in an element block of three-dimensional, two-node elements, the elements in the block will be treated as beam elements. The name, `beam_section_name`, can be used by the `SECTION` command line in a `PARAMETERS FOR BLOCK` command block.

One of several different types of beam cross section shapes may be specified with the `SECTION` command. The geometry of each section is defined by up to six geometric parameters, “d1” through “d6”. Some sections require all six parameters some as few as two. If a command of the form `D1 = <real>d1` is used the `d1` dimension will be constant for all elements in an element block. If a command of the form `D1 VARIABLE = <STRING>d1_var` the `d1` dimension will be read from the mesh file from the element variable named `d1_var`. When using the `DN VARIABLE` commands the dimension can be different on every element allowing modeling of tapered beams or other similar structures.

All dimensions used by the beam cross section must be defined in the input deck. If a given beam section geometry requires a certain “d” dimension then that dimension must be specified with exactly one `DN = command` or `DN VARIABLE = command`.

The following beam cross sectional geometries are supported:

- **ROD**: Solid circular or elliptical section. See Figure 6.8.
- **TUBE**: Thin walled circular or elliptical tube section. See Figure 6.9.
- **BAR**: Solid rectangular section. See Figure 6.10.
- **BOX**: Thin walled rectangular section. See Figure 6.11. `D3` is constant for all four walls.
- **I**: Standard symmetric I beam section. See Figure 6.12. Width of both flanges given by `D4` width of the web given by `D3`.
- **L**: Thin walled L section. See Figure 6.13.
- **C1**: Thin walled C section with open side oriented towards the s axis. See Figure 6.14.
- **C2**: Thin walled C section with open side oriented towards the t axis. See Figure 6.15.
- **T**: Thin walled T section with web oriented along the t axis. See Figure 6.16.
- **T1**: Thin walled T section with web oriented along the s axis. See Figure 6.17.
- **Z**: Thin walled Z section. See Figure 6.18.
- **HAT**: Thin walled hat section. See Figure 6.19. All plates of the same thickness `D4`.
- **I2**: Asymmetric I beam section. See Figure 6.20. The widths and thickness of the two flanges may be specified independently for this section.

Many sections (all but the **ROD** and **BAR**) have numerical integration schemes that assume thin walls. This means there is only a single integration point through the thickness of the beam flanges, plates, webs, walls, etc. If a thin walled beam section is defined with plate thickness that are relatively thick compared to the overall beam width and height the numerical properties of the beam may be inaccurate. Specifically the thin walled formulation will tend to underestimate the bending torsional stiffness and strength of thick walled beams.

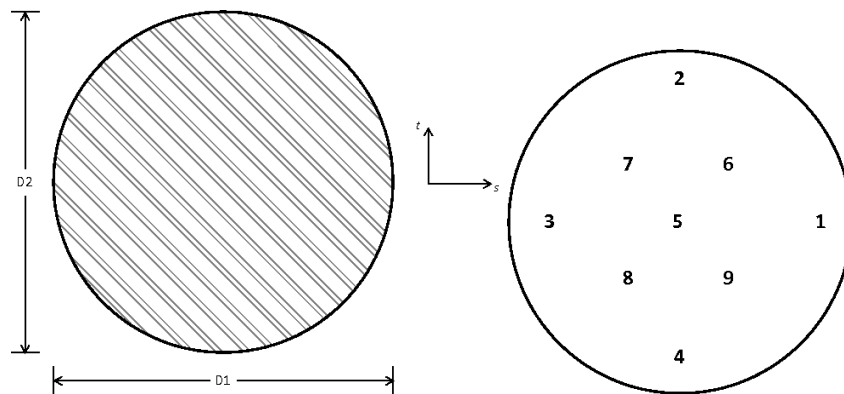


Figure 6.8: Beam ROD Cross Section

Most of the sections require the `T AXIS` or `T AXIS VARIABLE` command line. If the beam has a circular or tube cross section, and the width of the beam exactly equals the height, then the `T_AXIS` need not be specified. If the `T_AXIS` is not specified for one of these circularly symmetric cross sections the code will arbitrarily pick a t axis at each beam that is perpendicular to the beam.

Beam section parameters can be specified as constant for all beams in the section with commands

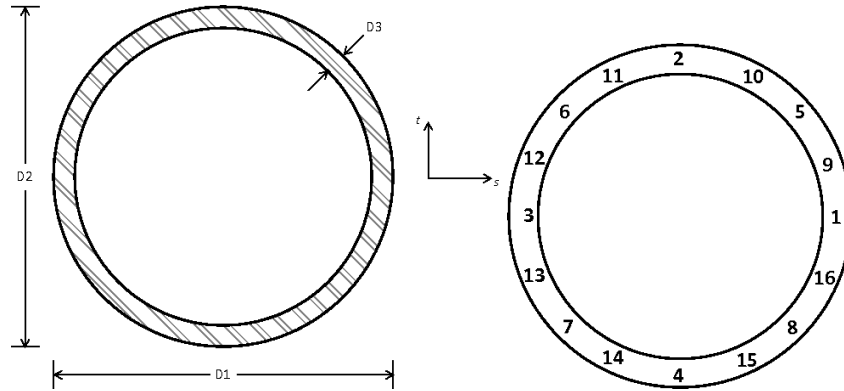


Figure 6.9: Beam TUBE Cross Section

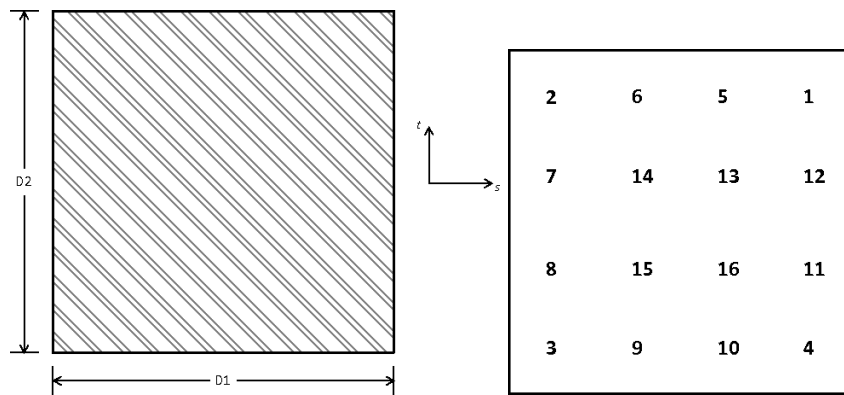


Figure 6.10: Beam BAR Cross Section

such as D3 or T AXIS. Alternatively a set of beam parameters that vary from element to element can be specified with variants of these commands with `VARIABLE` at the end, such as `D3 VARIABLE` or `T AXIS VARIABLE`. When the `VARIABLE` variants of commands are used, the command specifies the name of an attribute field on the input mesh that contains the parameter. For D1, D2, D3, D4, D5, and D6 the field should contain one entry per element. For `AXIS OFFSET VARIABLE` the field should contain either two or three entries per element. If the field for the axis offset contains two entries, it specifies the offset in the local s, t coordinate system. If it contains three entries, it specifies the offset in the global x, y, z coordinate system. For `T AXIS VARIABLE` the field should contain three entries per element, the vector components of the T axis at each element.

For beams it is necessary to specify a local Cartesian r, s , and t coordinate system for each beam. The r -axis lies along the length of the beam. The t -axis is specified by the user as a vector in the global coordinate system. If the t -axis provided by the user is not orthogonal to r then the code will alter t so that it is orthogonal to r as described below. The s -axis is computed from the cross product of the t -axis and the r -axis. The t -axis is then recomputed as the cross product of the r -axis and the s -axis to ensure that the t -axis is orthogonal to the r -axis. These local direction vectors are all normalized, so the user-input vectors do not have to be unit vectors.

If the initial position of the t -axis is to be parallel to the global Z-axis, then the command line

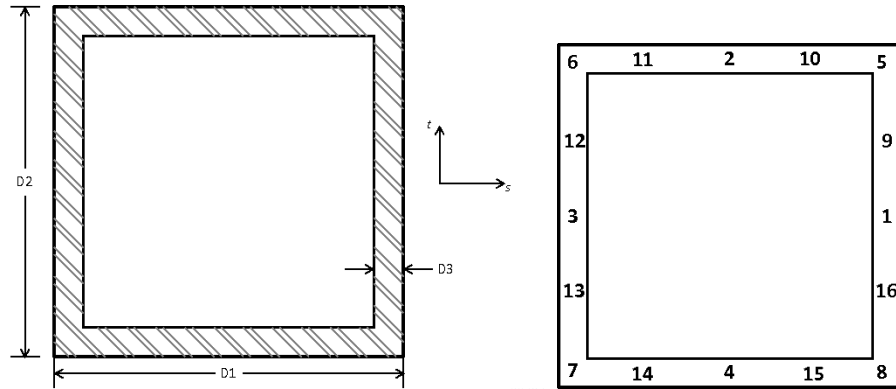


Figure 6.11: Beam BOX Cross Section

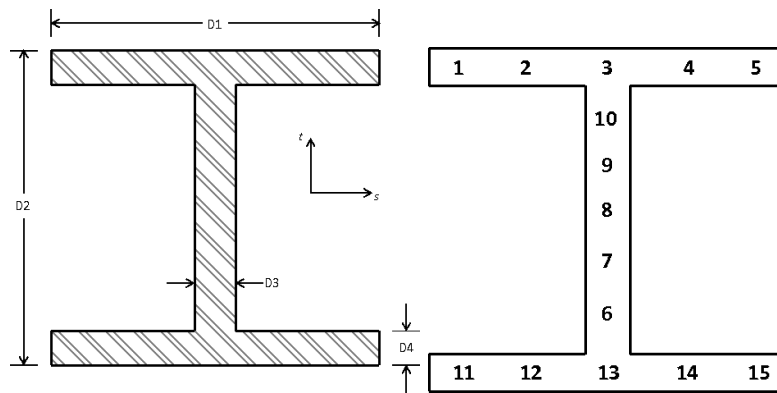


Figure 6.12: Beam I Cross Section

T AXIS = 0 0 1

should be used. If the initial position of the t -axis is to be parallel to a vector $(0.5, 0.8660, 0)$ in the global coordinate system, then the command line

T AXIS = 0.5 0.8660 0.0

should be used. The t -axis and s -axis must always be orthogonal to the beam r axis so that they will change position as the beam moves through space. This change in position is tracked internally by the computations for the beam element. Note that the t -axis given to the beam section also affects how the beam is oriented in lofted contact.

6.2.7.1 Reference Axis Offsets

By default, the r -axis coincides with the geometric center line of the beam. The geometric center line of the beam is defined by the location of the two nodes defining the beam connectivity. It is possible to offset the local r -axis, s -axis, and t -axis from the geometric center line of the beam. To do this, one can use one of commands the REFERENCE AXIS, AXIS OFFSET, AXIS OFFSET GLOBAL, or AXIS OFFSET VARIABLE. Only one of these commands may be used at a time.

The REFERENCE AXIS command line has the options CENTER, TOP, RIGHT, BOTTOM, and LEFT.

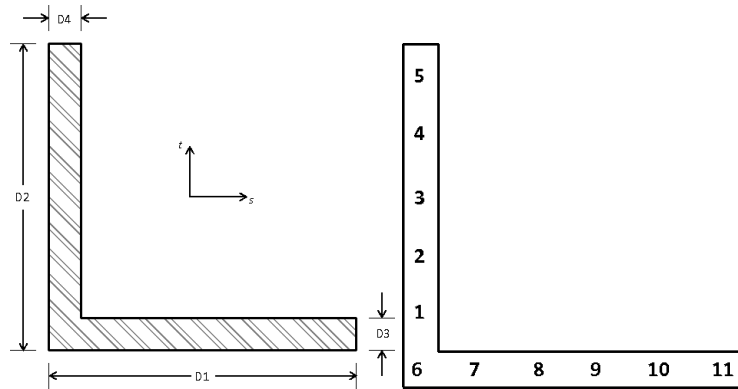


Figure 6.13: Beam L Cross Section

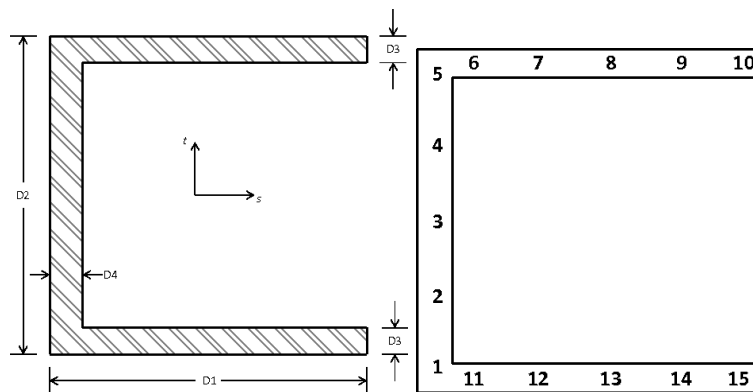


Figure 6.14: Beam C1 Cross Section

The `CENTER` option is the default, which means that the r -axis coincides with the centroid of the beam section. If the `TOP` option is used, the r -axis is moved to the top of the middle of the beam. `BOTTOM` corresponds to the bottom middle of the beam. `LEFT` corresponds to the left center of the beam. `RIGHT` corresponds to the right center of the beam. The location of the r -axis when using these options is shown for two beam sections in Figure 6.21. For all options, the s -axis and the t -axis remain parallel to their original positions before the translation of the r -axis.

Note for doubly symmetric cross sections the point `CENTER` always lies at the exact center of the cross section between the `LEFT` and `RIGHT` points and the `TOP` and `BOTTOM` points. For non doubly symmetric beam cross sections the beam centroid will depend on the particular beam dimensions and will move in relation to the `LEFT`, `RIGHT`, `TOP`, and `BOTTOM` points.

The `AXIS OFFSET` command line allows the user to offset the local coordinate system from the geometric center line by an arbitrary distance. The `s_offset` value shifts the r -axis along the beam local s -axis. The `t_offset` value shifts the r -axis along the beam local t -axis. The s , t , and r axes are only shifted, not rotated by this offset.

Alternatively, the axis offset can be specified in global coordinates using the `AXIS OFFSET GLOBAL` command. This command takes three parameters, which are the x , y , and z components of the offset in the global coordinate system. Only the portion of the offset that lies in the element

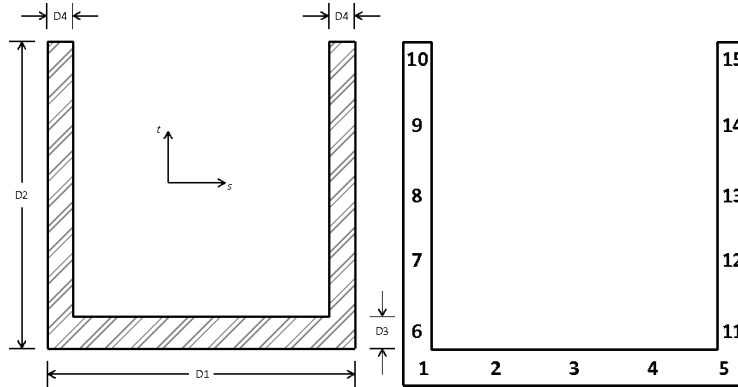


Figure 6.15: Beam C2 Cross Section

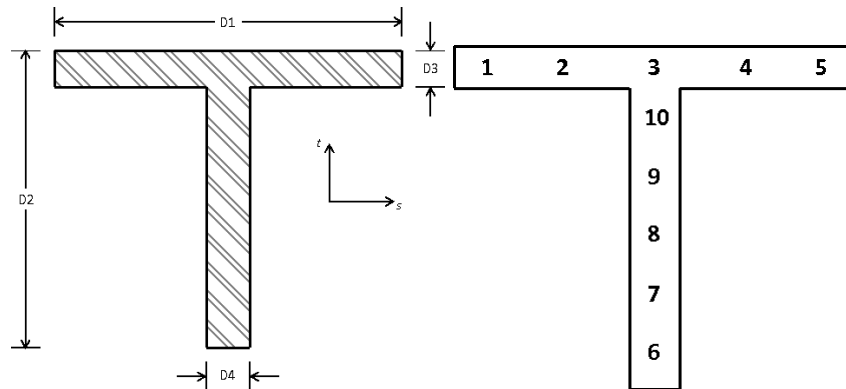


Figure 6.16: Beam T Cross Section

local s t plane will be used for the offset.

The `AXIS OFFSET VARIABLE` variable command behaves similar to the `AXIS OFFSET` command except that the offsets are specified element by element as defined by the input `axis_offset_var` field. This field should have two entries per element to define the element local s and t offsets.

6.2.7.2 Section Integration

Strains and stresses are computed at the midpoint of the two noded beam element beam. The integration of the stresses over the cross section at the midpoint is used to compute the internal forces in the beam. Each beam section has its own integration scheme.

The integration scheme for each of the sections is shown in Figure 6.8 through Figure 6.20. The numbers in these figures show the relative location of the integration points in the s t coordinate system.

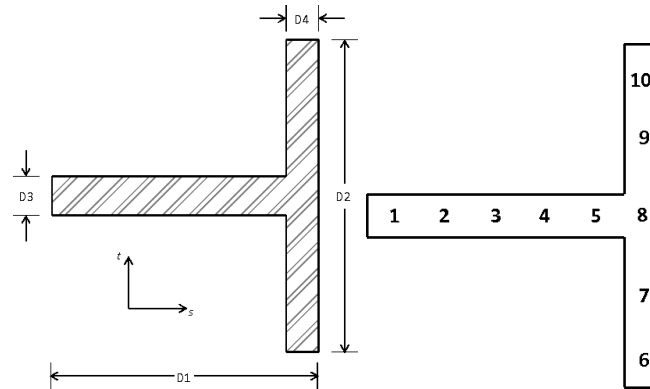


Figure 6.17: Beam T1 Cross Section

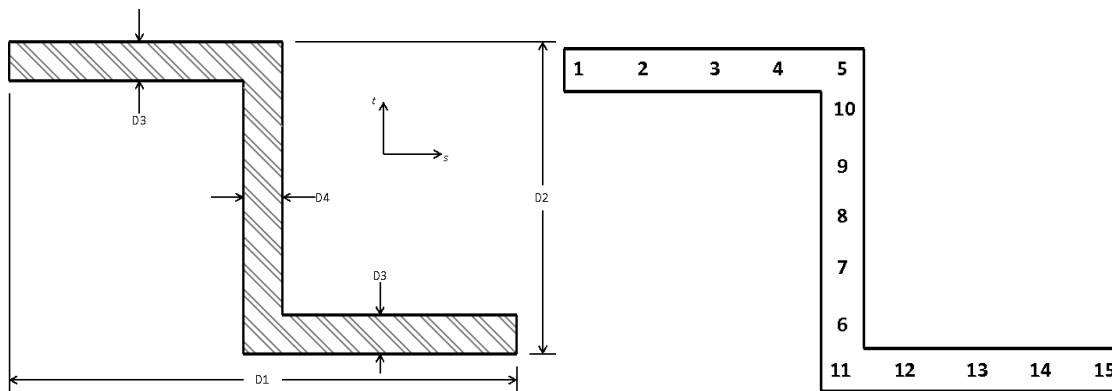


Figure 6.18: Beam Z Cross Section



Warning: Beam integration point shear stresses only account for torsional shear strains and do not contain effects due to transverse shear strains. Transverse shear forces are calculated from a linear elastic approximation using the given elastic constants and cross section. See the following discussion for more details.

At each integration point the beam element calculates strains and stresses that are related by the chosen material model. The formulation of the beam is like the typical Timoshenko theory and allows for transverse shear strains. However, in the Sierra/SM beam element the transverse shear is treated separately from the other strains. The axial stresses and strains are fully accounted for at the integration points, but the shear strains at the integration points are due only to the torsional deformation. In addition, the shear stress at the integration points is the circumferential torsional shear stress, with respect to the r axis, and the radial torsional shear stress is assumed to be zero (accurate for circular cross sections, an approximation for others). The transverse shear response is considered to be linear elastic, and the transverse shear contribution to the internal forces and moments is added apart from the contributions from the integration over the cross section. The linear elastic response is based on the user specified elastic constants.

Users should exercise caution when using the beam element with plasticity in problems that have large deformations or thick beams. Since the transverse shear response is only linear elastic, it will

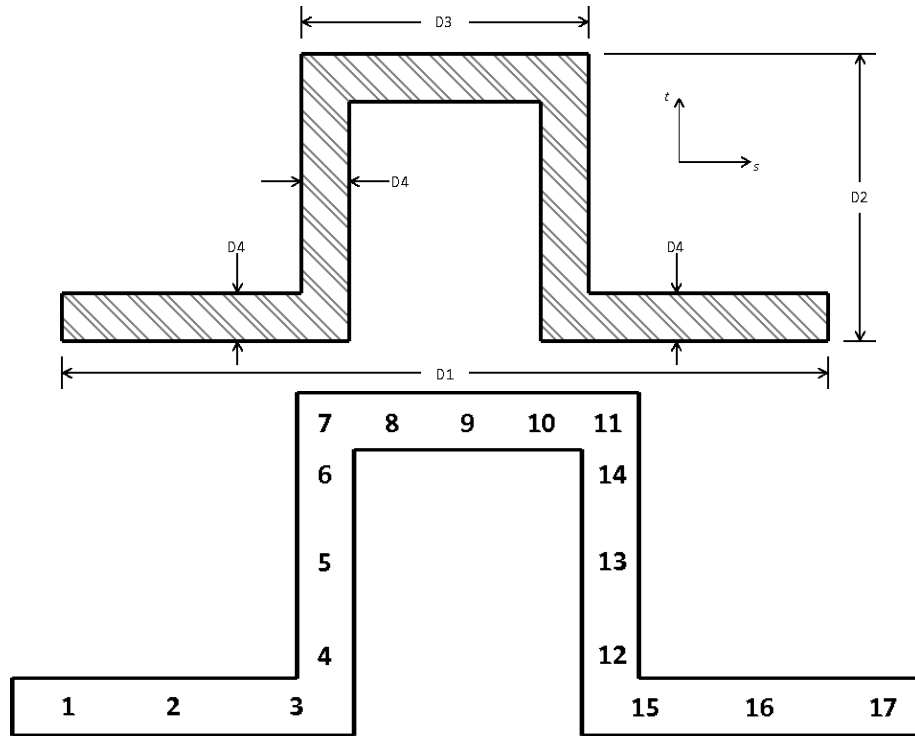


Figure 6.19: Beam HAT Cross Section

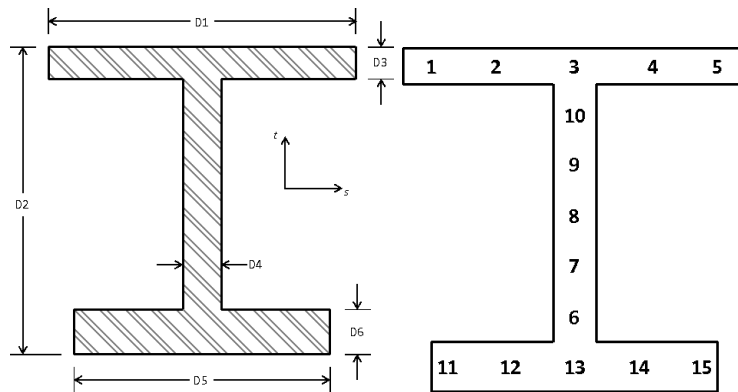


Figure 6.20: Beam I2 Cross Section

not properly account for other material models. Also, users should be careful to account for the integration point shear stresses only containing circumferential torsional shear stress.

Integration point data can be output for beam elements in much the same manner that it is requested for other elements by using commands in the `RESULTS OUTPUT` command block as described in Chapter 9. At each integration point, there are two components of strain and two components of stress. These components of stress and strain are in the axial and in-plane circumferential directions. The integration-point strain variables are named `beam_strain_axial` and `beam_strain_shear`, and the integration-point stress variables are `beam_stress_axial` and `beam_stress_shear`. In addition, state data is available at each integration point.

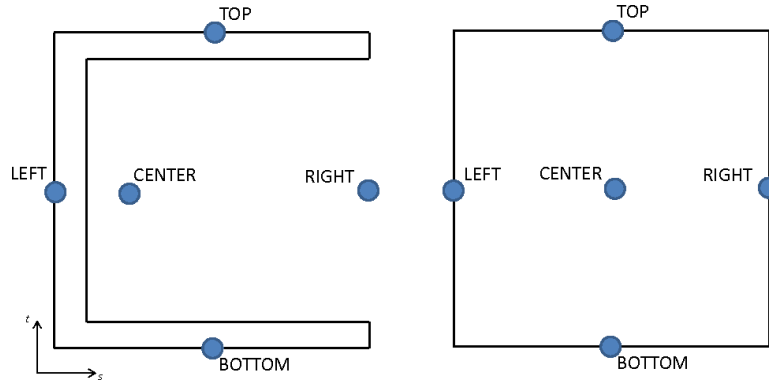


Figure 6.21: Locations of REFERENCE AXIS command options

An important aspect of integration point output for beam elements is that the data is output in arrays sized by the number of integration points for the chosen beam cross-section (for example 9 for a rod and 15 for an I).

The beam axial and shear strains and stresses are output in sequence with the integration point numbering. For example, the first value of `beam_strain_axial`, designated in the output as `_1` for a rod or `_01` for the other sections, is the axial strain at the first integration point. The numbering continues to the number of integration points for that beam section—up to `_9` for rod sections, `_15` for I sections, and `_16` for other sections.

An alternative method of accessing the stress is through the symmetric tensor `unrotated_stress`. Because a beam element uses local axes (i.e., a co-rotational system), `unrotated_stress` is given with respect to the r , s , and t coordinate system, and its `_xx` and `_xy` components are equivalent to the axial and torsional circumferential shear stresses, respectively, discussed above. Stress in general is stored as a symmetric tensor that contains six components, but for beams four of these components are never used. The axial stress at the first integration point is designated by `unrotated_stress_xx_01` (or `unrotated_stress_xx_1` for a rod section, with this pattern true for all integration point quantities discussed below) and the circumferential torsional shear stress at the first integration point is `unrotated_stress_xy_01`. The other four components, `unrotated_stress_yy_01`, `unrotated_stress_zz_01`, `unrotated_stress_yz_01` and `unrotated_stress_xz_01`, are unused and are set to zero. The stresses at other integration points are named `unrotated_stress_xx_NN` and `unrotated_stress_xy_NN`, where `NN` is a number from 01 to 16.

Optionally the lofted geometry and a visual representation of beam integration point locations may be included in the output mesh file with use of the `VISUALIZE INTEGRATION POINTS = ON` command. When the extra visualization is turned on each beam will be represented both by the original line element and also and by a set of brick elements. The brick elements will represent the shape of the beam cross section, there will be one brick element per beam integration point. The visualize integration points option can be helpful in ensuring that the beam is being setup as desired both in size, orientation, and offsets from the neutral axis.

When using the `VISUALIZE INTEGRATION POINTS` command some beam integration point quantities can be visualized directly on the lofted hexahedral geometry. Any field that is defined at

beam integration points will be copied out to the hexahedral representation elements. For example this includes fields such as:

- `intg_point`: Integration point number
- `intg_weight`: Integration point weight, should sum to 1.0 over all the integration points.
- `intg_shear_flow_dir`: Integration point shear flow direction
- `beam_stress_axial`: Axial component of beam stress
- `beam_stress_shear`: Shear component of beam stress
- `beam_strain_axial`: Axial component of beam strain
- `beam_strain_shear`: Shear component of beam strain
- `lame_state_<model_name>`: Material state variables.

It is possible to access mean values for the internal forces at the midpoint of the beam. The axial force at the midpoint of the beam is obtained by referencing the variable `beam_axial_force`. The transverse forces at the midpoint of the beam in the *s*-direction and the *t*-direction are obtained by referencing `beam_transverse_force_s` and `beam_transverse_force_t`, respectively. The torsion at the midpoint of the beam (the moment about the *r*-axis), is obtained by referencing `beam_moment_r`. The moments about the *s*-axis and the *t*-axis are obtained by referencing `beam_moment_s` and `beam_moment_t`, respectively.

Elements in a section can be made rigid by including the `RIGID BODY` command line. The `RIGID BODY` command line specifies an identifier that maps to a rigid body command block. See Section 6.3.1 for a full discussion of how to create rigid bodies and Section 6.3.1.1 for information on the use of the `RIGID BODIES FROM ATTRIBUTES` command.

Under certain conditions, the rotational mass of beams will be automatically scaled. This scaling is performed only when the computed time step from the axial modes is larger than the computed time step using the bending modes. This allows for larger critical time steps to be taken during the simulation. However, the user does have the option to turn it OFF via the `ROTATIONAL MASS SCALING` line command.



6.2.7.3 Beam Element Accuracy

It is worth noting that due to a finite accuracy integration rule the integration error in beam behavior is dependent on the loading case. For pure axial loading, the error is nearly zero throughout both the elastic and plastic regimes for all cross sections. This is due to axial resistance being a function of sectional area, which is explicitly computed from section properties.

However, in the bending cases there is varying levels in error dependent on section, bending direction, and loading value. In general, the error percentage is low (roughly 1 to 2%) for very elastic loadings in the *s*- and *t*- bending directions when strain is about 10^{-6} . Thin-walled sections with more integration points, such as the hat, are more accurate in bending in both the elastic and plastic regimes when bent in the *s*- and *t*-directions. Error can be larger (up to 10%) for elastic bending in directions of than the *s* and *t* axis. Additionally errors will tend to be larger (up to 10%) in the elastic plastic or fully plastic transition regimes.

Torsional loading for some cross sections can potentially produce large error. The integration assumes no out of plane warping during torsion. Thus the torsional resistance of sections that do have out of plane warping (open sections) may be greatly over estimated. For example, the circular rod

will see no sectional warping thus having less deviation from the analytical solution, particularly in the elastic regime. However, the hat cross section would see high amounts of sectional warping which is not fully accounted for in the solution produced by the code. In cases such as this, results will be vastly different than their analytical solutions. Additionally, it is important to point out that there is no bend-twist twist coupling for open sections, as this produces spurious unstable behavior.

6.2.8 Truss Section

```
BEGIN TRUSS SECTION <string>truss_section_name
  AREA = <real>cross_sectional_area
  DIAMETER = <real>diameter
  AREA MESH VARIABLE = <string>area_var_name
  INITIAL LOAD = <real>initial_load
  PERIOD = <real>period
  RIGID BODY = <string>rigid_body_name
  RIGID BODIES FROM ATTRIBUTES = <integer>first_id
    TO <integer>last_id
END [TRUSS SECTION <string>truss_section_name]
```

The `TRUSS SECTION` command block is used to specify the properties for a truss element. If this command block is referenced in an element block of three-dimensional, two-node elements, the elements in the block will be treated as truss elements. The parameter, `truss_section_name`, is user-defined and is referenced by a `SECTION` command line in a `PARAMETERS FOR BLOCK` command block.

The cross-sectional area for truss elements can be specified by the `AREA` command line. Alternatively the area can be set by defining the diameter for the cylindrical truss element. If specified the value `cross_sectional_area` or `diameter` will be used for all truss elements of the block element block. Alternatively the `AREA MESH VARIABLE` may be used to provide the name of a variable on the input genesis mesh that lists the element by element attributes. If the truss area is provided by an the element attribute the `area_var_name` should be set to “attribute”. Note one and only one of `AREA`, `DIAMETER`, or `AREA MESH VARIABLE` must be given for the truss section.

The truss can be given some initial load over some given time period. The magnitude of the load is specified by the `INITIAL LOAD` command line. If the load is compressive, the sign on the value `initial_load` should be negative; if the load is tensile, the sign on the value `initial_value` should be positive. The period is specified by the `PERIOD` command line.

The initial load is applied over some period by specifying the axial strain rate in the truss, $\dot{\epsilon}$, over some period p . At some given time t , the strain rate is

$$\dot{\epsilon} = \frac{ap}{2} [1 - \cos(\pi t/p)], \quad (6.3)$$

where

$$a = \frac{2F_i}{EAp}. \quad (6.4)$$

In Equation (6.4), F_i is the initial load, E is the modulus of elasticity for the truss, and A is the area of the truss. Over the period p , the total strain increment generates the desired initial load in the truss.

During the initial load period, the time increments should be reasonably small so that the integration of $\dot{\epsilon}$ over the period is accurate. The period should be set long enough so that if the model was held in a steady state after time p , there would only be a small amount of oscillation in the load in the truss.

Certain boundary conditions should not be activated until after the pre-stressing is done. During the pre-stressing, time-independent boundary conditions such as fixed displacement will most likely be turned on. Time-dependent boundary conditions such as prescribed acceleration or prescribed force will most likely be activated after the pre-stressing is complete.

Elements in a section can be made rigid by including the `RIGID BODY` command line. The `RIGID BODY` command line specifies an indenter that maps to a rigid body command block. See Section 6.3.1 for a full discussion of how to create rigid bodies and Section 6.3.1.1 for information on the use of the `RIGID BODIES FROM ATTRIBUTES` command.



Warning: Truss elements currently do not work correctly as a rigid body due to lack of cross-section geometry to define the moment of inertia about the truss axis. They should be used only if the user explicitly includes the command line `INERTIA =` in the `RIGID BODY` command block to specify the mass moment of inertia for the entire rigid body. (See Section 6.3.1.) The recommendation is to use beam elements for rigid bodies rather than truss elements if possible.

Axial stress, which is the only stress component for a truss element, is output as a symmetric tensor with six components to be compatible with the notion of volumetric stress. Because of this, for every truss element, six values of stress are stored and output, but only the first value is ever used. The axial stress is therefore output as `stress_xx` and the other five stress components, `stress_yy`, `stress_zz`, `stress_xy`, `stress_yz`, `stress_zx` are all unused and set to zero.

6.2.9 Spring Section

```
BEGIN SPRING SECTION <string>spring_section_name
  FORCE STRAIN FUNCTION = <string>force_strain_function
  DEFAULT STIFFNESS <real>default_stiffness
  PRELOAD = <real>preload_value
  PRELOAD DURATION = <real>preload_duration
  RESET INITIAL LENGTH AFTER PRELOAD = <string>NO|YES
  MASS PER UNIT LENGTH = <real>mass_per_unit_length
END [SPRING SECTION <string>spring_section_name]
```

The `SPRING SECTION` command block is used to specify the properties for a spring element. If this command block is referenced in an element block of three-dimensional, two node elements, the elements in the block will be treated as spring elements. The parameter, `spring_section_name`, is user-defined and is referenced by a `SECTION` command line in a `PARAMETERS FOR BLOCK`

command block.

The spring behavior is governed by the force-engineering strain function which is specified by the `FORCE STRAIN FUNCTION` command line. The force generated by the spring element is based on the evaluation of the user specified `force_strain_function`, which has units of force. The strain is defined by $(L_i - L_0)/L_0$ with L_i the current spring length and L_0 the original spring length. This allows the force-strain function to be length independent.

Positive values of the force strain function correspond to tension and negative values to compression. The force-strain function should be defined over the full range of strains that the spring may see, both positive and negative. The following are examples of possible spring stiffness functions.

```
BEGIN FUNCTION FORCE_STRAIN_2
  # "x" is the signed input strain value
  # This is a linear spring with stiffness of 1000
  TYPE = ANALYTIC
  EVALUATE EXPRESSION = "1000*x"
END

BEGIN FUNCTION FORCE_STRAIN_3
  # Nonlinear "chain" element. Has stiffness in tension only
  TYPE = PIECEWISE LINEAR
  BEGIN VALUES
    0      0
    5  5000
  END
END

BEGIN FUNCTION FORCE_STRAIN_1
  # Nonlinear complex spring with gaps and variable stiffness
  TYPE = PIECEWISE LINEAR
  BEGIN VALUES
    -0.5 -1000
    -0.1   0
    0      0
    0.1  1000
    0.2  5000
    0.5 10000
  END
END
```

The `DEFAULT STIFFNESS` command block specifies the spring stiffness used during the first time step and during preload. In all other situations the spring stiffness is based on the slope of the force-strain function evaluated at the previous time step and the current time step. The unit of `default_stiffness` is force.

To specify a preload on the spring both the `PRELOAD` and `PRELOAD DURATION` command line must be specified. The `PRELOAD` command line specifies the magnitude of the preload force, while the `PRELOAD DURATION` command line specifies how long the preload application should take, in seconds.

An optional preload input, `RESET INITIAL LENGTH AFTER PRELOAD`, is used when the user would like the initial length of the spring to reset to the displaced length after the preload has occurred. If this command line is not specified, or is set to `NO`, the initial length of the spring is the undeformed length as calculated from the input mesh.

Springs can have a mass as defined by the `MASS PER UNIT LENGTH` command line. When defining the spring section this value must be specified. A zero mass spring is allowed by setting `MASS PER UNIT LENGTH` explicitly to zero. Note, in explicit computations if the spring has no mass the spring has no defined element time step. For a massless spring the effect of the spring on the global time step will be factored into the nodal [3.2.3.1](#) or Lanczos [3.2.1.5](#) time step estimates.



6.2.10 Damper Section

```
BEGIN DAMPER SECTION <string>damper_section_name
  AREA = <real>damper_cross_sectional_area
END [DAMPER SECTION <string>damper_section_name]
```

The `DAMPER SECTION` command block is used to specify the properties for a damper element. If this command block is referenced in an element block of three-dimensional, two-node elements, the elements in the block will be treated as damper elements. The parameter, `damper_section_name`, is user-defined and is referenced by a `SECTION` command line in a `PARAMETERS FOR BLOCK` command block.

The cross-sectional area for damper elements is specified by the `DAMPER AREA` command line. The value `damper_cross_sectional_area` is the cross-sectional area of the dampers in the element block.

The damper area is used only to generate mass associated with the damper element. The mass is the density for the damper element multiplied by the original volume of the element (original length multiplied by the damper area).

The force generated by the damper element depends on the relative velocity along the current direction vector for the damper element. If $\mathbf{n}$ is a unit normal pointing in the direction from node 1 to node 2, if $\mathbf{v}_1$ and $\mathbf{v}_2$ are the velocity vectors at nodes 1 and 2, respectively, then the force generated by the damper element is

$$F_d = \eta \mathbf{n} \cdot (\mathbf{v}_2 - \mathbf{v}_1), \quad (6.5)$$

where η is the damping parameter. Currently, the damping parameter must be specified by using an elastic material model for the damper element. The value for Young's modulus in the elastic material model is used for the damping parameter η .



6.2.11 Point Mass Section

```
BEGIN POINT MASS SECTION <string>pointmass_section_name
```

```

VOLUME = <real>volume
MASS = <real>mass
IXX = <real>Ixx
IYY = <real>Iyy
IZZ = <real>Izz
IXY = <real>Ixy
IXZ = <real>Ixz
IYZ = <real>Iyz
RIGID BODY = <string>rigid_body_name
RIGID BODIES FROM ATTRIBUTES = <integer>first_id
    TO <integer>last_id
MASS VARIABLE = <string>mass_variable_name
INERTIA VARIABLE = <string>inertia_variable_name
OFFSET VARIABLE = <string>offset_variable_name
ATTRIBUTES VARIABLE NAME = <string>attrib_variable_name
END [POINT MASS SECTION <string>pointmass_section_name]

```

A point mass element is simply a mass at a node, which can be a convenient modeling tool in certain instances. The user can create an element block with one or more point masses. A point mass is a sphere element attached to a single node. A point mass will have its mass added to the mass at the connected node. (Other mass at the node will be derived from mass due to elements attached to the node.) The mass at a node due to a point mass is treated like any other mass at a node derived from an element. The mass due to point mass will be included in body force calculations and kinetic energy calculations, for example.

Point masses are a convenient modeling tool to be used in conjunction with rigid bodies. An element block including one or more point masses can be included in a collection of element blocks used to define a rigid body. The element block of point masses can be used to adjust the total mass and inertia properties for the rigid body. (The point mass element does not have to be used only in conjunction with rigid bodies. One can place a point mass at a node associated with solid or structural elements.)

An element block in which the connectivity for each element contains only one node can be used as a collection of point masses. This command block would have the following form:

```

BEGIN PARAMETERS FOR BLOCK <string>block_id
    MATERIAL = <string>material_name
    MODEL     = <string> material_model_name
    SECTION = <string>point_mass_section_name
END PARAMETERS FOR BLOCK <string>block_id

```

The element block associated with the point mass must reference a material command block just like any other element block. If the point mass section specifies a volume, then the product of the density specified in the material block and the volume specified in the section block is used to calculate the element mass.

The `VOLUME` command specifies the volume of a point mass element. The mass of the element is then found by multiplying that volume by the material model density.

The `MASS` command explicitly specifies the mass.

The `IXX`, `IYY`, `IZZ`, `IXY`, `IXZ`, `IYZ` commands are used to explicitly define the symmetric inertia tensor entries for the point mass. Note that currently, the trace of the inertial tensor is added to the nodal rotational mass and the rest of the inertia tensor entries are ignored.

The `OFFSET` is used to define the offset of the mass with respect to the connected node. Currently the offset affects only the rotational properties of the point mass. The rotational mass of the point mass is increased by mass times offset distance squared.

The `MASS VARIABLE` specifies that the mass of the point mass elements is to be read from the input mesh file from the variable with the specified name (use the name "attribute" to read the mass from the mesh block attributes.)

The `INERTIA VARIABLE` specifies that the inertia tensor of the point mass elements is to be read from the input mesh file from the variable with the specified name (use the name "attribute" to read the inertia tensor from the mesh block attributes.) When this command is used, the mesh file variable should contain six entries per element, ordered as follows: `Ixx`, `Iyy`, `Izz`, `Ixy`, `Ixz`, `Iyz`.

The `OFFSET VARIABLE` specifies that the offset vector of the point mass elements is to be read from the input mesh file from the variable with the specified name (use the name "attribute" to read the offset vector from the mesh block attributes.) When this command is used, the mesh file variable should contain three entries per element, ordered as follows: `Offset_x`, `Offset_y`, `Offset_z`.

The `ATTRIBUTES VARIABLE NAME` specifies that all properties of the point mass element are to be read from the input mesh file of the variable with the specified name (use the name "attribute" to read from the mesh block attributes.) When this command is used, the mesh file variable should contain ten entries per element, ordered as follows: `Mass`, `Ixx`, `Iyy`, `Izz`, `Ixy`, `Ixz`, `Iyz`, `Offset_x`, `Offset_y`, `Offset_z`.

Multiple ways are provided to specify the raw attributes of mass, inertia and offset for the point mass element. The user must be careful to only use one of these ways to define each property. Using multiple ways to define the mass will result in errors or warnings because of ambiguities.

If the user has access to the SEACAS codes from Sandia National Laboratories, The codes in this library may be used to generate element blocks with point mass elements. See Reference [10] for an overview of the SEACAS codes. By using various SEACAS codes, an element block with one or more point masses may be generated. For each point mass in the element block, an eight-noded hexahedral element centered at the point mass location should be created. The hexahedron may have arbitrary dimensions, but it is best to work with a unit cube. For example, to generate a point mass at coordinates (13.5, 27.0, 3.1415), a $1 \times 1 \times 1$ (unit) hexahedron centered at (13.5, 27.0, 3.1415) should be created. The SEACAS program SPHGEN3D may be used to convert the hexahedron to a zero-dimensional element located at the center of the hexahedron in three-dimensional space. In the provided example, SPHGEN3D would create a zero-dimensional, single-noded element at coordinates (13.5, 27.0, 3.1415).

Elements in a section can be made rigid by including the `RIGID BODY` command line. The `RIGID BODY` command line specifies an indenter that maps to a rigid body command block. See Section 6.3.1 for a full discussion of how to create rigid bodies and Section 6.3.1.1 for information on the use of the `RIGID BODIES FROM ATTRIBUTES` command.



6.2.12 Particle Section

```
BEGIN PARTICLE SECTION <string>sph_section_name
  RADIUS MESH VARIABLE = <string>var_name|<string>attribute|
    SPHERE INITIAL RADIUS = <real>rad
  RADIUS TIME STEP = <string>time
  PROBLEM DIMENSION = <integer>1|2|3(3)
  CONSTANT SPHERE RADIUS
  FINAL RADIUS MULTIPLICATION FACTOR = <real>factor(1.0)
  FORMULATION = <string>MASS_PARTICLE|SPH(SPH)
  MONAGHAN EPSILON = <real>monaghan_epsilon(0.0)
  MONAGHAN N = <real>monaghan_n(0.0)
  SPH ALPHAQ PARAMETER = <real>alpha(1.0)
  SPH BETAQ PARAMETER = <real>beta(2.0)
  DENSITY FORMULATION = <string>MATERIAL|KERNEL(MATERIAL)
  MAXIMUM RADIUS CHANGE = <real>minVal <real>maxVal
END [PARTICLE SECTION <string>sph_section_name]
```

SPH (smoothed particle hydrodynamics) is useful for modeling fluids or for modeling materials that undergo extremely large distortions. One must be careful when using SPH for modeling. SPH tends to exhibit both accuracy and stability problems, particularly in tension. An SPH particle interacts with other nearest-neighbor SPH particles based on radius properties of all the elements involved; SPH particles react with other elements, such as tetrahedra, hexahedra, and shells, through contact. Many more details on the theoretical background of SPH are available in [11].

Contact interactions between the particles in a particle element block and other element types may be defined using the `CONTACT NODE SET` option described in Section 8.4.2. All particles in the particle element block must be included in a node set if the `CONTACT NODE SET` option is used.

The particle section should only be associated with elements with the `SPHERE` topology. Sphere elements are point elements that are connected to only a single node.

All the particles contained in a particle element block must be given some initial radius. There are two options for setting the initial radius for each particle. First, each particle can be given the same radius. To set the radius for each particle in an element block to the same value, use the `SPHERE INITIAL RADIUS` command line. The parameter `rad` on this command line sets the radius for all the particles in the element block.

Alternatively, the radius for each particle can be read from a mesh file. The radii can be read from a variable on the mesh file from attributes associated with the element block. To read some variable from the mesh file for the radii, the command

```
RADIUS MESH VARIABLE = sph_radius
```

may be used, where `sph_radius` is the variable name on the mesh file. To use the variable associated with a specific time on the mesh file, the `RADIUS TIME STEP` command line should be used to select the specific time. To read the attributes associated with the particles, the command line

```
RADIUS MESH VARIABLE = attribute
```

should be inserted (as shown) into the `PARTICLE SECTION` command block. Pronto3D [1] only offers the attribute option. The attribute option should be used to compare Sierra/SM and Pronto3D results.

If the attribute option is used, each particle should have two attributes defined per element. The first attribute is the particle radius, and the second attribute is the particle volume. The most common way to generate a particle mesh is to first generate a solid element mesh and then use the utility 'sphgen3d' to convert the solid elements into sphere elements. Sphgen3d will convert each solid element into a sphere and place it at the source element centroid. In the output mesh file generated by sphgen3d The radius attribute of the element blocks will be proportional to the source element size and the volume attribute will be the volume of the source element.

The `FINAL RADIUS MULTIPLICATION FACTOR` command may be used to increase or decrease the initial particle radii. The factor specified will scale the particle radii input with either a constant or a mesh file attribute.

After the radii are initialized, the user may specify whether the radii are to remain constant or are to change throughout the analysis. The `CONSTANT SPHERE RADIUS` command line is an optional command line that prevents the sphere radius from changing over the course of the calculation. By default, the sphere radii will expand or contract based on the changing density in the elements to satisfy the relation that element mass (a constant) equals element volume times element density. If the `CONSTANT SPHERE RADIUS` command line appears, then the radii for all particles will remain constant.

The `DENSITY FORMULATION` command controls the method used to update the density, and thus SPH radii, during the analysis. The default `MATERIAL` option uses the material density and nodal mass to compute a volume for each particle at a given time. The radius is then computed as the cube root of that volume. The alternative `KERNEL` option computes the current density for each particle using the mass of that particle and the SPH kernel density function. The `KERNEL` option may be necessary when large expansions of particles are expected (such as when modeling large density changes in gases). The `MATERIAL` option generally changes particle densities, and thus radii, less than the `KERNEL` option, so is appropriate for analyses that do not have large density fluctuations.

The `MAXIMUM RADIUS CHANGE` command may be used to control how much a given SPH particle is allowed to expand or shrink over the course of the analysis. For example, setting

```
MAXIMUM RADIUS CHANGE = 0.2 10.0
```

would indicate that the radius of any particle cannot become less than 20% or more than 1000% of the particle's original radius. By default, the particle can shrink or expand as much as its density change requires. This option may be useful, for example, in explosive modeling if the particle radii are expanding excessively causing analysis performance and stability problems.

An analysis using SPH may be inherently one-, two-, or three-dimensional. Dimensionality of the problem may be specified using the `PROBLEM DIMENSION` command line. The possible value for this command line is 1 (one-dimensional), 2 (two-dimensional), or 3 (three-dimensional). The default value is 3 for three-dimensional. The internal SPH kernel functions are modified depending on the value set on the `PROBLEM DIMENSION` command line. If, for example, a problem is

inherently two-dimensional in nature, more accurate results may be obtained by specifying the dimension of the problem as 2.

The `FORMULATION` command defines the SPH formulation to use, and has the following options:

- `SPH`, general smoothed particle hydrodynamics.
- `MASS_PARTICLE`, Particles only have mass, and no stiffness. Particles do not interact with each other in any way.

The `MONAGHAN EPSILON` and `MONAGHAN N` commands set the Monaghan viscosity coefficients. Monaghan viscosity may allow the SPH algorithm to behave better, particularly in tension. Standard values to use are 0.2 for epsilon and 4 for n. See (Reference [12]) for more information on the usage of these parameters.

The `SPH ALPHAQ PARAMETER` and `SPH BETAQ PARAMETER` commands control the SPH viscosity terms. Larger values lead to more viscosity. Alpha is associated with a viscosity parameter that is proportional to the dilatational modulus of the material. Beta is associated with a viscosity parameter that is proportional to the change in material density over a time step.

Utility commands. In addition to the SPH-related command lines described above (which appear in the `PARTICLE SECTION` command block), the `BEGIN SPH OPTIONS` is another SPH-related command block:

```
BEGIN SPH OPTIONS
  SPH SYMMETRY PLANE <string>+X|+Y|+Z|-X|-Y|-Z
    <real>position_on_axis(0.0)
  SPH DECOUPLE STRAINS: <string>material1 <string>material2
END [SPH OPTIONS]
```

If used, this command block should be placed in the Presto region scope (in contrast to most SPH-related commands which are in the `PARTICLE SECTION` command block). The symmetry conditions are applied to all SPH element blocks.

The `SPH SYMMETRY PLANE` command line may be used to reduce model sizes by specifying symmetry planes and modeling only a portion of the model. Due to the non local nature of SPH element integration, symmetry planes cannot be defined with boundary conditions alone; these planes must be explicitly defined. A plane is defined by an outward normal vector aligned with one of the axes (+X, +Y, +Z, -X, -Y, -Z) and some point on the axis, which represents a point in the plane. Suppose for example, the outward normal to the plane of symmetry is in the negative Y-direction (-Y) and the plane of symmetry passes through the y-axis at $y = +2.56$. Then the definition for the symmetry plane would be:

```
SPH SYMMETRY PLANE -Y +2.56
```

The `SPH DECOUPLE STRAINS` command line prevents two dissimilar materials from directly interacting. Generally, the material properties at a particle are the average of the material properties from nearby particles. If particles with very dissimilar material properties are interacting, this interaction can create problems. The `SPH DECOUPLE STRAINS` command line ensures that particles with very dissimilar material properties do not directly interact by material-property averaging,

but instead interact with a contact-like interaction. The two material types that are not to interact are specified by the parameters `material1` and `material2`. These parameters will appear as material names on a `MATERIAL` command block.

Conversion to particles at initialization. To facilitate the creation of particle-based meshes, an additional SPH-related command block is the `BEGIN CONVERSION TO PARTICLES AT INITIALIZATION` block:

```
BEGIN CONVERSION TO PARTICLES AT INITIALIZATION
  BLOCK = <string list>block_names
  SECTION = <string>particle_section
END [CONVERSION TO PARTICLES AT INITIALIZATION]
```

This block, which must be placed in the Presto region scope, specifies that an input mesh consisting of *solid elements* be converted into particles at initialization. Particle elements are created at the centroid of each element with appropriate mass and volume, and the initial solid elements of the input mesh are deleted. Any initial and boundary conditions are inherited from the corresponding initial solid element; furthermore, the particle will be associated with the same node set or side set which *any node* of the parent element belongs to.

Only elements within the blocks specified by the `BLOCK` command will be converted to particles. Each new element will be assigned an arbitrary, but new and unique global numerical ID.

The `SECTION` command must specify a user-defined section of `SPH`, `RKPM` or `PERIDYNAMICS` type; it is important to note that the `SECTION` command originally associated with the initial solid element-block command block(s) will therefore be disregarded. It is worth noting that a particle section should *not* be specified in the initial solid element-block command block(s), since particle sections are *not* compatible with solid element input mesh topologies; instead, a compatible solid element section should be specified in the initial element-block definition(s).

Display. For purposes of visualizing the element stresses, it may be necessary to copy these element variables into nodal variables. This can easily be done by defining a `USER VARIABLE` command block (Section 11.2.4) in conjunction with a `USER OUTPUT` command block (Section 9.3.2). Once the nodal variable is defined, it can be output in a `RESULTS OUTPUT` command block (Section 9.3.1). An example is provided below. The `SPH` element blocks for the problem are element blocks 20, 21, and 22. All other element blocks are non-SPH elements.

- In the region scope:

```
# Define the variable
BEGIN USER VARIABLE nodal_stress
  TYPE = NODE SYM_TENSOR LENGTH = 1
END
#
# Copy element stress to nodal stress
BEGIN USER OUTPUT
  BLOCK = block_20 block_21 block_22
  COPY ELEMENT VARIABLE stress TO NODAL VARIABLE nodal_stress
END
```



```
# Output nodal stress
BEGIN RESULTS OUTPUT output_presto
  DATABASE NAME = sph.e
  DATABASE TYPE = exodusII
  AT TIME 0.0 INCREMENT = 1.0e-04
  NODAL nodal_stress
END RESULTS OUTPUT output_presto
```



Known Issue: For problems with rotations, the SPH algorithm generates rotational spring like forces on the boundary because of known deficiencies in the algorithm in distinguishing between rigid body rotation and deformation. The SPH algorithm does not conserve rotational or angular momentum.



Known Issue: If a BEGIN PRESSURE (Section 7.6.1) block is used with the CONVERSION TO PARTICLES AT INITIALIZATION command, the pressure is not applied. Pressure can be applied to particles by using the ELEMENT DEATH command block (Section 6.5).



6.2.13 Peridynamics Section

```
BEGIN PERIDYNAMICS SECTION <string>section_name
  MATERIAL MODEL FORMULATION = <string>CLASSICAL|
    PERIDYNAMIC_BOND_BASED|PERIDYNAMIC_STATE_BASED
  HORIZON = <real>horizon
  INFLUENCE FUNCTION = <string>influence_function
  BOND DAMAGE MODEL = <string>CRITICAL_STRETCH|
    CRITICAL_STRETCH_DECAY|CRITICAL_STRETCH_PERVASIVE_DAMAGE|
    ELEMENT_VARIABLE|NONE(NONE)
    [<real>critical_stretch] [<real>damage_coefficient]
    [<real>min_allowable_multiplier] [<real>C1] [<real>C2]
    [<real>C3]
  HOURGLASS STIFFNESS = <real>stiffness(0.0)
  BOND CUTTING BLOCK = <string list>block_names
  STRESS DECAY FUNCTION = <string>stress_decay_function
  BOND VISUALIZATION = <string>OFF|ON(OFF)
  SHORT RANGE FORCES = OFF|ON(OFF) [<real>staticCoefficient(15.0)]
    [<real>dampingCoefficient(1.0)]
    [<real>thresholdAlphaDistance(1.35)]
    [<real>thresholdBetaDistance(0.9)]
END [PERIDYNAMICS SECTION <string>section_name]
```

Peridynamics is an extension of classical solid mechanics that allows for the modeling of bodies in which discontinuities occur spontaneously. For this reason, peridynamics is well suited for modeling phenomena such as cracking. This section describes the PERIDYNAMICS SECTION

command block. Details regarding peridynamics methodology and suggested usage guidelines can be found in Section 10.

A `PERIDYNAMICS SECTION` should only be associated with elements with the `SPHERE` topology. Sphere elements are point elements that are connected to only a single node. Sphere elements may be created, for example, using the `spheregen.py` utility (preferred) or the `sphgen3d` utility.



Warning: The default parallel decomposition for models containing sphere elements is typically poor. It is recommended that an initial rebalance be carried out for parallel analyses with peridynamics. See Section 6.8 for details regarding the `INITIAL REBALANCE` command.

Within a `PERIDYNAMICS SECTION`, the `MATERIAL MODEL FORMULATION` command line specifies the application of either a state-based peridynamics material model, a bond-based peridynamics material model, or a classical material model adapted for use with peridynamics as described in Section 10. Currently, only the `ELASTIC` material model is available with the `PERIDYNAMIC_STATE_BASED` material model formulation, and only the `ELASTIC` and `ELASTIC_PLASTIC` material models are available with the `PERIDYNAMIC_BOND_BASED` material model formulation. Setting `MATERIAL MODEL FORMULATION = CLASSICAL` enables the interface between Sierra/SM peridynamics and the `LAME` material library. This interface allows the material models described in Section 5.2 to be used with peridynamics.

The peridynamic horizon is set with the `HORIZON` line command. As described in Section 10, the horizon determines the region of nonlocality for a given peridynamic element. A peridynamic element interacts directly with all peridynamic elements that are within a distance equal to the horizon in the reference configuration. The horizon may be specified as either a constant value or as a multiple of the element radius. By default, the command applies a constant horizon equal to the specified value of `horizon` to all elements associated with the given peridynamics section.

A user-defined peridynamic influence function may be defined using the `INFLUENCE FUNCTION` command line. The influence function provides a means for controlling the relative influence of a neighbor within the family of a given peridynamic element as a function of the neighbor's distance from the element. Distance from the element is normalized by the horizon and ranges from zero to one. A value of zero corresponds to a distance of zero from the element, and a value of one corresponds to a distance from the element equal to the horizon. By default, the influence function is set to:

$$\omega = \begin{cases} 1 & \text{for } 0 \leq x < \frac{1}{2} \\ 4x - 4x^2 & \text{for } \frac{1}{2} \leq x \leq 1, \end{cases} \quad (6.6)$$

which results in the influence function having a value of 1.0 for all neighbors within a distance of half the horizon, and a value that drops off parabolically beginning at half the horizon and ending with a value of zero at edge of the horizon. (An influence function that drops off smoothly near the edge of the horizon tends to mitigate numerical noise.) An arbitrary influence function may be defined with the `INFLUENCE FUNCTION` command line by providing a function name corresponding to a `DEFINITION OF FUNCTION` command block in the Sierra scope (see Section 2.1.5). The input to this function is the distance in the reference configuration from an element to a given

neighbor, divided by that element's horizon. The output is the weighting value, ω . Influence functions do not apply to bond-based peridynamic material models.

The bond damage model is specified by the `BOND DAMAGE MODEL` command. The available options for `BOND DAMAGE MODEL` are `CRITICAL_STRETCH`, `CRITICAL_STRETCH_DECAY`, `CRITICAL_STRETCH_PERVASIVE_DAMAGE`, `ELEMENT_VARIABLE`, and `NONE`. The default option, `NONE`, disables bond breaking. The critical stretch bond breaking law is activated by specifying `CRITICAL_STRETCH`. If this model is selected, the user must specify `critical_stretch`, the value of stretch at which bonds are broken, where stretch is computed as bond elongation divided by the initial bond length. The `CRITICAL_STRETCH_DECAY` bond breaking law provides a modification to the standard `CRITICAL_STRETCH` law. In the case of `CRITICAL_STRETCH_DECAY`, the value of critical stretch is modified based on the value of peridynamic damage as follows:

$$s_{\text{crit}} = s_{\text{crit}_0} \max(m, 1 - c d_{\text{max}}), \quad (6.7)$$

where s_{crit} is the critical stretch, s_{crit_0} is the nominal critical stretch, m is the minimum allowable multiplier, c is the damage coefficient, and d_{max} is the peridynamic damage. The value of peridynamic damage, d_{max} , is the maximum peridynamic damage (percentage of broken bonds) for the two elements connected by the given bond. An additional modification to the standard `CRITICAL_STRETCH` law is available through the `CRITICAL_STRETCH_PERVASIVE_DAMAGE` bond breaking law. In this case, the critical stretch is computed as follows,

$$s_{\text{crit}} = s_{\text{crit}_0} \min \left(C3, 1 + C2 \frac{d_{\text{max}} - C1}{1 - d_{\text{max}}} \right), \quad (6.8)$$

where $C1$, $C2$, and $C3$ are user-supplied coefficients. The `ELEMENT_VARIABLE` option allows bonds to be broken based on the maximum value of a specified element variable (e.g., an element variable associated with a material model, such as equivalent plastic strain). In the case of a bond that connects two peridynamics elements for which the specified element variable is defined, the average of the two element values is compared against the threshold value to determine bond breakage. If the specified element variable is defined on only one of the two elements connected by a given bond, the single available value is compared against the threshold value.

Hourglass stiffness for peridynamics is controlled with the `HOURLASS STIFFNESS` command. Within the peridynamics hourglass stiffness routine, the parameter `stiffness` is multiplied by a material's bulk modulus to determine the resistance to deformation in potential zero-energy modes. By default, no hourglass stiffness is computed for peridynamics; hourglass stiffness must be activated by including a `HOURLASS STIFFNESS` command line. Note that peridynamics hourglass control is completely independent from the hourglass control applied to under-integrated elements (Section 6.1.6.6).

By default, a peridynamic bond is formed between a given peridynamic element and all peridynamic elements within its horizon. Sometimes this results in the formation of bonds that are unphysical. For example, bonds may be incorrectly formed between two components that are within a distance equal to the horizon in the reference configuration but that are not physically connected. The bond cutting feature provides a mechanism for controlling the formation of bonds. The `BOND CUTTING BLOCK` command designates blocks in the `block_names` list as bond cutting blocks. Any bond that would pass through a surface of a bond cutting block is omitted from

the analysis. For example, to prevent the creation of bonds between two components that are in close proximity but are not physically connected, a user may include a block in the input mesh file that lies between these components and designate that block as a bond cutting block. The bond cutting block may be any type of block (e.g., a shell, hexahedral, or tetrahedral mesh), provided that Sierra/SM is capable of skinning it (see Section 8.4). If a sphere element mesh for use with peridynamics has been created from a hexahedral mesh (for example, with `spheregen.py`), it may be convenient to simply include the original hexahedral element mesh as a bond cutting block. Any block designated as a bond cutting block is automatically removed from the analysis following the initialization of bonds.

The optional `STRESS DECAY FUNCTION` command line specifies a function by which the computed stress from a classical material model is decayed as a function of peridynamic damage. The use of stress decay function prevents an unphysical increase in the magnitude of pairwise forces as the number of broken bonds increases for a given element. By default, the stress decay function is:

$$f_d = \begin{cases} 1.0 - 1.25d & \text{for } 0 \leq d < 0.8 \\ 0 & \text{for } 0.8 \leq d \leq 1.0, \end{cases} \quad (6.9)$$

where d is the peridynamic damage (percentage of broken bonds). Note that the `STRESS DECAY FUNCTION` command line has an effect only when `MATERIAL MODEL FORMULATION` is set to `CLASSICAL` and is ignored when either `PERIDYNAMIC_STATE_BASED` or `PERIDYNAMIC_BOND_BASED` material models are in use.

The `BOND VISUALIZATION` command provides a mechanism for visualizing bonds through the representation of bonds as bar elements in the output database. This feature is set to `OFF` by default and may be activated with the command `BOND VISUALIZATION = ON`. The bond visualization feature is intended as a verification tool for mesh creation and is not recommended for used for production runs.

The `SHORT RANGE FORCES` command provides a contact like formulation to prevent interpenetration of peridynamics nodes that do not have bonds. By default these forces are off.



Warning: Setting `BOND VISUALIZATION = ON` results in the creation of additional elements in the output file. Because each bond is represented by an additional element, the resulting output file may grow in size by several orders of magnitude relative to the standard output file. For this reason, `BOND VISUALIZATION` is recommended only as a mesh verification tool for small- to medium-sized problems and should not be used for production runs.



Known Issue: Peridynamics is not compatible with several modeling capabilities. Peridynamics is currently not compatible with restart. Also, the only type of rebalancing that can be done with peridynamics is an initial rebalance.



6.2.14 User Mount Section

```
BEGIN USER MOUNT SECTION <string>user_mount_section_name
```

```

MOUNT TYPE = FAIL_TRUSS|SPRING_DASH_POT|SNUBBER|
            TORSIONAL_SPRING|6_DOF_LINEAR_SPRING|USER_SUBROUTINE
PARAMETERS = <real list>data
PARAMETERS VARIABLE = <string>attribute_name
T AXIS = <real> <real> <real>
T AXIS VARIABLE <string>attribute_name
S AXIS = <real> <real> <real>
S AXIS VARIABLE <string>attribute_name
ZERO LENGTH R AXIS = <real> <real> <real>

MOUNT FORCE SUBROUTINE = <string>mountForceSub
MOUNT INIT  SUBROUTINE = <string>mountInitSub
MOUNT SIZE  SUBROUTINE = <string>mountSizeSub
END [USER MOUNT SECTION <string>user_mount_section_name]

```

The `USER MOUNT SECTION` command block provides a capability to link two nodes with an arbitrary nonlinear connection. If this command block is referenced in an element block of three-dimensional, two node elements, the elements in the block will be treated as user mount elements. The name, `user_mount_section_name`, can be used by the `SECTION` command line in a `PARAMETERS FOR BLOCK` command block.

The `MOUNT TYPE` is command is used to specify a small number of `USER MOUNTS` that have been implemented directly in the code rather than through user subroutines. Note that the user mounts were originally developed for use in Naval applications and are also called Navy Mounts (Nmounts). When defining a mount via user subroutine (the typical use case), specify `MOUNT TYPE = USER_SUBROUTINE`.

User mount parameters can be specified for all mounts in an element block with the `PARAMETERS` command. `PARAMETERS` specifies the user defined parameters for this mount, constant for all elements that use this section. The number of parameters should match the number required by the specified `MOUNT TYPE`; if the wrong number of parameters are provided an error is issued. Orientation dependent parameters may need re-ordering given a different initial orientation (see below).

Alternatively, the `PARAMETERS` command, along with the optional `AXIS` commands discussed below, has a variant with `VARIABLE` that allows the setting of mount parameters that vary from element to element. When the `VARIABLE` variants of commands are used, the command specifies the name of an attribute field on the input mesh that contains the parameter(s). These attributes should be sized appropriate to the `MOUNT TYPE` when `PARAMETERS VARIABLE` is used. Similarly, an attribute specified by `T AXIS VARIABLE` must contain three entries per element, the vector components of the t -axis at each element.

Optionally, the `T AXIS` and `S AXIS` commands are used to specify an initial orientation of the user mount in a local Cartesian r , s , and t coordinate system (`T AXIS VARIABLE` and `S AXIS VARIABLE` should be used when the initial orientation varies by element). The r -axis lies along the length of the user mount. Either the s -axis or the t -axis can be specified as a vector in the global coordinate system. If the axis provided is not orthogonal to r then the code will alter it so that it is orthogonal to r as described below. The t -axis is computed from the cross product of the s -axis and

the r -axis. The s -axis is then recomputed as the cross product of the r -axis and the t -axis to ensure that the s -axis is orthogonal to the r -axis. These local direction vectors are all normalized, so the input vectors do not have to be unit vectors. If the initial position of the t -axis is to be parallel to the global Z-axis, the command line

```
T AXIS = 0 0 1
```

should be used.

The s -axis and t -axis are orthogonal to the user mount r -axis, so they will change position as the user mount moves through space. This change in position is tracked internally by the computations for the element. If neither a s -axis nor a t -axis is specified, a default one will be used, following the above rule-set.

The `ZERO LENGTH R AXIS` command can be used to explicitly specify the r -axis. By default the user mount r axis aligns with the bar element. The `ZERO LENGTH R AXIS` will override this default so that the R axis is always in the given constant direction. This command is particularly useful for zero length mounts where the bar aligned r -axis is ill-defined. If a mount has approximately zero length, and the `ZERO LENGTH R AXIS` command is not given the global X-axis will be used for the r axis.

6.2.14.1 User Mount API

The `MOUNT FORCE|INIT|SIZE SUBROUTINE` commands can be used to define dynamically linked mount response functions. See Chapter 11 for more information on building and executing user subroutines. The API for these specific subroutines is given below.

6.2.14.2 Coordinate Frame

Typically mount interaction routines are written entirely within the mount element local coordinate frame. The frame is shown in Figure 6.22.

- X_{elem} is the normalized vector from node 1 to node 2. This may change as the structure deforms.
- V is the user provided orientation vector. This vector must not be completely parallel to X_{elem} in the initial configuration.
- Z_{elem} is the normalized cross product of X_{elem} and V
- Y_{elem} is the normalized cross product of Z_{elem} and X_{elem}

The direction cosines are available as part of the API, should the mount require them. Note that the frame may rotate as the structure deforms. Zero length coordinate systems can pose particular problems. At this time, such mounts require an associated coordinate frame, and do not rotate with the structure. Note that there are several definitions of the element coordinate frame. In this API a convention with the y element direction in the same plane as the orientation vector, v , and the beam axis is adopted.

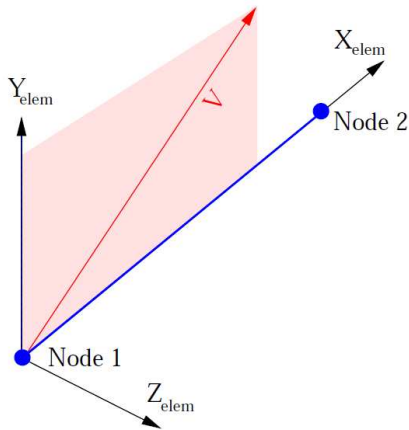


Figure 6.22: User Mount Coordinate System

6.2.14.3 Mount Output Variables

The user mount defines a number of specific element output variables.

- `user_mount_r_axis`, `user_mount_s_axis`, `user_mount_t_axis`: vector quantities that can be used to visualize the user mount coordinate system ensuring it is setup and rotated as expected.
- `element_state_variables`: state variables used for the mount computations. The state variable array size and meaning will vary based on the user subroutine calls and/or mount type

6.2.14.4 Mount Sizing Routine

The host code must allocate memory in which to read the parameters associated with each mount and the state variables used by the mount. For a given mount type these sizes must be constant, but the size can vary on different types of mounts. The mount sizing routine is used to define these sizes and has the following definition.

SUBROUTINE SIZE_MOUNT:

INPUT DATA:
NONE

OUTPUT DATA:
 INTEGER nParam: The number of parameters used to define each
 mount. For example gap length in a gap element

 INTEGER nState: The number of state variables used by the
 mount.

INTEGER nActive: An indicator of which degrees of freedom are active. This integer is a logical sum of contributions from each dof, where X=1, Y=10, Z=100, RotX=1000, RotY=10000, RotZ=100000. Thus, a mount that couples only translations has nActive=111, while a mount with only rotations is 111000. A mount with all six is 111111.

RESTRICTIONS:

For a given mount type the sizing routine must always return the same number of parameters and number of state variables. Note, This routine may be called by the host code at any time, for example at initialization or immediately prior to writing a restart file.

6.2.14.5 Mount Initialization Routine

The mount initialize routine provides an opportunity for the mount to set its initial state data. The host code will call the mount initialize routine once at the start of the run. The mount initialize routine will be called on each mount element individually.

SUBROUTINE INITIALIZE_MOUNT:

INPUT DATA:

REAL startTime: The analysis time at the start of the first analysis load step or time step.

REAL X_Elem(3): Mount inputs and outputs will be defined in a local coordinate system computed by the host code. X_Elem is a vector aligned with the mount element. Y_elem and Z_elem are orthogonal to X_Elem. The origin of the element coordinate system is located at the first node of the mount element, with X_Elem as the unit vector of the element coordinate system in global XYZ coordinates. Based on mount configuration at start of analysis.

REAL Y_Elem(3): Unit vector of the element coordinate system in global XYZ coordinates. Based on mount configuration at start of analysis.

REAL Z_Elem(3): Unit vector of the element coordinate system in global XYZ coordinates. Based on mount configuration at start of analysis.

REAL coord1(6): The element coordinates of node 1 on the mount at

the beginning of the analysis. Order is
(translational_x, translational_y, translational_z,
rotational_x, rotational_y, rotational_z)

REAL coord2(6): The element coordinates of node 2 on the mount at
the beginning of the analysis.

REAL param(nParam): The parameters associated with the current mount
element. These parameters may be set in the
input deck or read from the mesh file
attributes. The number of parameters read by
the host code and passed to this routine is the
same 'nParam' value that is defined by the
'SIZE_MOUNT' routine.

OUTPUT DATA:

REAL sNew(nState): The initial mount state variables at the start of
the analysis. The number of state variables
passed out is the same 'nState' value that is
defined by the 'SIZE_MOUNT' routine.

REAL Z_ElemOut(3): The X_Elem of the local coordinate system will
always be mount aligned. Element frame is always
orthogonal. The host code will compute a valid
element coordinate system and pass it to the
initialization routine. The mount initialize
routine may modify the initial layout of the
element coordinate system by passing back a new
Z_Elem in Z_ElemOut. Z_ElemOut is defined in
global XYZ coordinates. Note, the Z_Elem used by
the host code must be orthogonal to X_Elem. If
Z_ElemOut is not orthogonal to X_Elem the host
code will update it such that: Y_Elem =
Cross(Z_ElemOut, X_Elem), and Z_Elem =
Cross(X_Elem, Y_Elem)

INTEGER errorCode: If the mount routine completed without hitting
any errors the errorCode should be set to zero. A
positive errorCode indicates the mount routine
hit a fatal error. If a mount returns a fatal
error the host code will report the mount ID and
errorCode in the log file and then abort
execution gracefully as soon as possible. A
negative errorCode indicates the mount hit a non
fatal warning. If a mount returns a warning code
the host code will report the mount ID and
errorCode in the log file and continue execution.

RESTRICTIONS:

- a) The mount initialize routine may not modify in any way any of the input arguments. Any such changes will at best be ignored and at worst may corrupt the host code's database causing unpredictable results.
- b) The mount initialize routine must never call abort or stop, if the mount encounters an error it must return an error code.
- c) The mount must explicitly set all output variables. All sNew values should always be set to something. The mount should always set the errorCode to something (zero if routine executed without any errors). If the mount just uses the default local coordinate system, initialize routines should copy ``Orient`` into ``OrientOut``.
- d) The mount initialize routine must not make any parallel calls and should not be aware that it is running in parallel or running on a specific processor (the host code may move it from one processor to another during the run).
- e) Reading or writing to/from any files within the mount code is highly discouraged. If done the file should be opened and closed each call rather than left open.
- f) The mount must not store any recoverable information in any location other than the state variables that are passed out of the INITIALIZE MOUNT routine. For example the mount routine should not set and then retrieve a static variable or attempt to do its own memory allocation and data storage.

6.2.14.6 Mount Execution Routine

The host code will call the mount execute routine any time the mount force must be evaluated. The mount execute routine takes as input the state of the mount at the beginning of a load step or time step. The execute routine returns the force in the mount during the load step. The execute routine should also update any internal mount state variables to the end of load step values. The mount execute routine will be called on each mount element individually. Note the mount execute routine may be called several times during a load step, for example for use in an iterative solver.

SUBROUTINE COMPUTE_MOUNT_FORCE:

INPUT DATA:

REAL startTime: The analysis time at the start of the current analysis load step or time step.

REAL deltaT: The length of the current analysis time step or load step. Note, the code may call the mount execute routine in order to probe the mount stiffness, if that is the case the deltaT passed in for the probe may be 0.0.

REAL X_Elem(3): Mount inputs and outputs will be defined in a local element coordinate system. X is a vector aligned with the mount element. Y and Z are orthogonal to X. The origin of the element coordinate system is located at the first node of the mount element. The host code will update the element coordinate system if the mount element undergoes rotation. Vectors defined in the mount configuration at end of load step.

REAL Y_Elem(3): Unit Y vector of the element coordinate system in global XYZ coordinates. Based on mount configuration at end of load step.

REAL Z_Elem(3): Unit Z vector of the local coordinate system in global XYZ coordinates. Based on mount configuration at end of load step.

REAL coord1(6): The element coordinates of node 1 on the mount at the end of the load step. Order is (translational_x, translational_y, translational_z, rotational_x, rotational_y, rotational_z)

REAL coord2(6): The element coordinates of node 2 on the mount at the end of the load step.

REAL disp1(6): The element displacement of node 1 at the end of the load step.

REAL disp2(6): The element displacement of node 2 at the end of the load step.

REAL vel1(6): The element velocity of node 1 at the middle of the load step.

REAL vel2(6): The element velocity of node 2 at the middle of the load step.

REAL param(nParam): The parameters associated with the current mount element. These parameters may be set in the input deck or read from the mesh file

attributes. The number of parameters read by the host code and passed to this routine is the same 'nParam' value that is defined by the 'SIZE_MOUNT' routine.

REAL sOld(nState): The mount state variables at the start of the load step. The number of state variables passed in is the same 'nState' value that is defined by the 'SIZE_MOUNT' routine.

OUTPUT DATA:

REAL force1(6): The element coordinate forces and moments at node 1 at the end of the load step.

REAL force2(6): The element coordinate forces and moments at node 2 during the load step.

REAL sNew(nState): The updated mount state variables at the end of the load step. The number of state variables passed out is the same 'nState' value that is defined by the 'SIZE_MOUNT' routine.

INTEGER errorCode: If the mount routine completed without hitting any errors the errorCode should be set to zero. A positive errorCode indicates the mount routine hit a fatal error. If a mount returns a fatal error the host code will report the mount ID and errorCode to the user then abort execution gracefully as soon as possible. A negative errorCode indicates the mount hit a non fatal warning. If a mount returns a warning code the host code will report the mount ID and errorCode to the user and continue execution.

RESTRICTIONS:

- a) The mount execute routine may not modify in any way any of the input arguments. Any such changes will at best be ignored and at worst may corrupt the host code's database causing unpredictable results.
- b) The mount execute routine must never call abort or stop, if the mount encounters an error it must return an error code.
- c) The mount must explicitly set all output variables. The return force should be set to zero if there is no force. sOld values should be copied into sNew if the mount state variables were not

changed. The mount should always set the errorCode to something (zero if routine executed without any errors).

- d) The mount execute routine must not make any parallel calls and should not be aware that it is running in parallel or running on a specific processor (the host code may move it from one processor to another during the run).
- e) Reading or writing to/from any files within the mount code is highly discouraged.
- f) The mount must not store any recoverable information in any location other than the state variables that are passed into and out of the COMPUTE MOUNT FORCE routine. For example the mount routine should not set and then retrieve a static variable or attempt to do its own memory allocation and data storage.

6.2.14.7 Example

The following is an example of use of the API for defining a simple mount. This example is written in Fortran 77. The mount designed here is a truss element with a failure criterion. The truss element behaves as a linear spring. If the strain in the truss becomes too large the truss breaks. A broken truss can never again support tensile or compressive loads even if later the strain is reduced below the break strain value.

```
//=====
      subroutine fail_truss_size(int nParam, int nState, int nActive)
      int nParam
      int nState
c
c Param(1) is the stiffness of the truss, expressed in force per strain
c Param(2) is the critical value of tensile strain at which the truss
c breaks.
c
      nParam = 2
c
c State(1) is the initial length of the truss State(2) indicates if the
c truss is broken or not, 1 is intact, 2 is broken
c
      nState = 2
c Truss does not couple to rotation,
      nActive = 111
      end
//=====
      subroutine fail_truss_initialize(startTime,
&                                     X_Elem, Y_Elem, Z_Elem,
&                                     coord1, coord2, param,
```

```

        &                                sNew, Z_ElemOut, errorCode)
c
c Input arguments
c
    double startTime
    double X_Elem(3), Y_Elem(3), Z_Elem(3)
    double coord1(6), coord2(6)
    double param(2)
c
c Output arguments
c
    double sNew(2)
    double Z_ElemOut(3)
    int errorCode
c
c Compute and store initial truss length, this is just the length in R
c Set the truss as initially unbroken
c
    sNew(1) = fabs(coord1(1)-coord2(1))
    sNew(2) = 1
c
c If truss is initially zero length, problem ill defined, abort
c
    if(sNew(1) == 0.0) then
        errorCode = 1
        return
    endif
c
c Use default element coordinate system
c
    Z_ElemOut(1) = Z_Elem(1)
    Z_ElemOut(2) = Z_Elem(2)
    Z_ElemOut(3) = Z_Elem(3)
    errorCode = 0
    end

//=====
subroutine fail_truss_compute(startTime, deltaT,
    &                                X_Elem, Y_Elem, Z_Elem,
    &                                coord1, coord2,
    &                                disp1, disp2,
    &                                vel1, vel2,
    &                                param,
    &                                sOld,
    &                                force1, force2,
    &                                sNew,
    &                                errorCode)
c

```

```

c Input arguments
c
    double startTime, deltaT
    double X_Elem(3), Y_Elem(3), Z_Elem(3)
    double coord1(6), coord2(6)
    double disp1(6), disp2(6)
    double vel1(6), vel2(6)
    double param(2)
    double sOld(2)

c
c Output arguments
c
    double force1(6), force2(6)
    double sNew(2)
    int errorCode

c
c Local variables
c
    double newLen
    double curStrain
    double curForce

c
c Check if truss is 'broken' if so it no force
c
    if(sOld(2) == 2) then
        force1(1) = 0.0
        force1(2) = 0.0
        force1(3) = 0.0
        force1(4) = 0.0
        force1(5) = 0.0
        force1(6) = 0.0
        force2(1) = 0.0
        force2(2) = 0.0
        force2(3) = 0.0
        force2(4) = 0.0
        force2(5) = 0.0
        force2(6) = 0.0
        sNew(1) = sOld(1)
        sNew(2) = sOld(2)
        errorCode = 0
        return
    endif

c
c Compute the new truss length, this is just the length in R
c
    newLen = fabs(coord1(1)- coord2(1))
c

```

```

c If length compressed to zero, element is invalid, make host code abort
c
    if(newLen == 0.0) then
        errorCode = 1
        return
    endif
c
c Compute the current strain as new length over original length.
c Compute tensile (positive) or compressive (negative) force based on
c stiffness time strain. force is applied along R with now off axis
c component. If strain exceeds critical value break the truss
c
    curStrain = (newLen/sOld(1))
    if(curStrain > param(2)) then
        sNew(2) = 2
        force1(1) = 0.0
        force1(2) = 0.0
        force1(3) = 0.0
        force1(4) = 0.0
        force1(5) = 0.0
        force1(6) = 0.0
        force2(1) = 0.0
        force2(2) = 0.0
        force2(3) = 0.0
        force2(4) = 0.0
        force2(5) = 0.0
        force2(6) = 0.0
        sNew(1) = sOld(1)
    else
        if(curStrain > 1.0) then
            curForce = (curStrain-1.0) * param(1)
        else
            curForce = -(1.0/curStrain) * param(1)
        endif
        force1(1) = curForce
        force1(2) = 0.0
        force1(3) = 0.0
        force1(4) = 0.0
        force1(5) = 0.0
        force1(6) = 0.0
        force2(1) = -curForce
        force2(2) = 0.0
        force2(3) = 0.0
        force2(4) = 0.0
        force2(5) = 0.0
        force2(6) = 0.0
        sNew(1) = sOld(1)
    end

```



```

        sNew(2) = sOld(2)
    endif
    errorCode = 0
end

```

6.2.15 Cylindrical Joint

```

BEGIN CYLINDRICAL JOINT SECTION <string>cylj_section_name
  AXIAL DIRECTION = <vector_3D> axial_direction
  axial direction = 0.0 0.0 -1.0
  AXIAL ROTATION STIFFNESS = <string>theta_stiffness
  AXIAL TRANSLATION STIFFNESS = <string>axial_stiffness
  INNER SURFACE = <string>inner_surface
  OUTER SURFACE = <string>outer_surface
  RADIAL ROTATION STIFFNESS = <string>yaw_stiffness
  RADIAL TRANSLATION STIFFNESS = <string>radial_stiffness
END [CYLINDRICAL JOINT SECTION <string> truss_section_name]

```

The CYLINDRICAL JOINT SECTION command block provides the ability to add nonlinear stiffness against the relative average motion of two surfaces (*inner_surface* and *outer_surface*) using a cylindrical system assumption. (In contrast, to kinematically prescribe a cylindrical boundary condition a user should use the CYLINDRICAL AXIS line command within a prescribed kinematic boundary condition command block, e.g., for prescribed displacement see [7.4.2.2](#).) The cylindrical joint capability is intended for modeling joints with cylindrical constraints (e.g., a disc mounted on a post) and it works best on joints that are axisymmetric or nearly so. In addition, unspecified stiffnesses are assumed to be zero for these joints so that free motion may be specified (e.g., a disc free to rotate on a post which has zero AXIAL ROTATION STIFFNESS, while all other stiffnesses are specified).

This command block should be referenced in an element block of a single three-dimensional two node element (i.e., a bar or beam element in typical meshing tools). The two node element is used to define the reference axis against which the relative motion of the two surfaces is computed. The element variables for this section type are listed in [6.5](#) and are stored on the single two node element.

The element variables *cylj_rst_origin*, *cylj_r_axis*, *cylj_s_axis*, and *cylj_t_axis* can be used to visualize the local co-rotational Cartesian reference axes, which is constructed in much the same way as for beam or bar elements. The value of *cylj_rst_origin* is at the center of the two node element and the *cylj_r_axis* is in the direction from first node to the second node as defined by the two node element connectivity (note that the *cylj_r_axis* is *not* the radial direction). The directions *cylj_s_axis* and *cylj_t_axis* are internally computed and are orthogonal to each other and *cylj_r_axis*.

The parameter, *cylj_section_name* is user-defined and is referenced by a SECTION command line in a PARAMETERS FOR BLOCK command block.

The parameter, *axial_direction* can optionally be input to define the direction along the axis of the cylindrical joint. Sierra/SM will take this direction and determine if the normal of the beam

element needs to be swapped to the opposite direction. If the command is not present, the beam normal will be used.

The function defined with `theta_stiffness` on the command line `AXIAL ROTATION STIFFNESS` is used to determine moment applied as a function of the two surfaces' relative motion illustrated in 6.23.

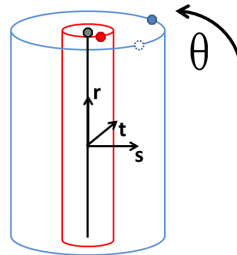


Figure 6.23: Cylindrical Joint Axial Rotation

The function defined with `axial_stiffness` on the command line `AXIAL TRANSLATION STIFFNESS` is used to determine force applied as a function of the two surfaces' relative motion illustrated in 6.24.

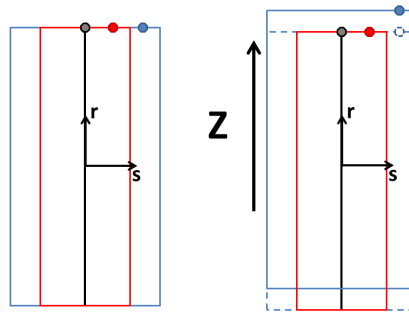


Figure 6.24: Cylindrical Joint Axial Translation

The function defined with `yaw_stiffness` on the command line `RADIAL ROTATIONAL STIFFNESS` is used to determine moment applied as a function of the two surfaces' relative motion illustrated in 6.25. This function should generally start at (0,0) and increase monotonically positive X positive Y quadrant.

The function defined with `radial_stiffness` on the command line `RADIAL TRANSLATION STIFFNESS` is used to determine force applied as a function of the relative motion illustrated in 6.26. This function should generally start at (0,0) and increase monotonically positive X positive Y quadrant.

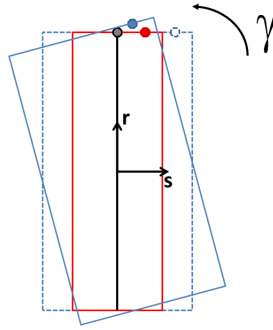


Figure 6.25: Cylindrical Joint Radial Rotation

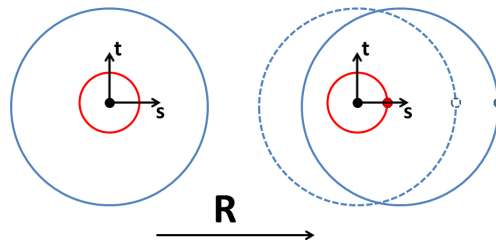


Figure 6.26: Cylindrical Joint Radial Translation

The `INNER SURFACE` and `OUTER SURFACE` commands specify the surfaces where relative motion is measured and the resulting force and moments are distributed. In a given time step, the averaged step motion in the reference Cartesian coordinate system is computed on the `inner_surface` and `outer_surface`. The accumulated translational and rotational motion is stored in the three-component vector element variables `cylj_rst_coord1` and `cylj_rst_theta1`, respectively, for the `inner_surface`. Similarly, the accumulated motion for `outer_surface` is stored in `cylj_rst_coord2` and `cylj_rst_theta2`.

The relative motion used for evaluating the stiffness functions is calculated by the vector math of $(\text{cylj\_rst\_coord2} - \text{cylj\_rst\_coord1})$ and $(\text{cylj\_rst\_theta2} - \text{cylj\_rst\_theta1})$. The relative motion of the `coord` values is used with the translational stiffness and the resulting force is stored in the element variable `cylj_rst_force`. Similarly, the `theta` values use the rotational stiffness and result in a moment stored in `cylj_rst_moment`. The element internal force and moment values are then applied equally and of opposite sign to the `inner_surface` and `outer_surface` and are distributed across each surface's nodes for the cylindrical joint element's internal force.

For output the “x” vector components of the `coord`, `theta`, `force`, and `moment` element variables correspond to the axial direction `cylj_r_axis`. The “y” and “z” components correspond to the `cylj_s_axis` and `cylj_t_axis` radial directions respectively, which are treated with the same

translational and rotational stiffness due to the axisymmetric assumption.

Table 6.5: Variables For Cylindrical Joint

Variable Name	Type	Comments
cylj_rst_origin	Vector_3D	(Reference axis origin)
cylj_r_axis	Vector_3D	(Reference axis r direction)
cylj_s_axis	Vector_3D	(Reference axis s direction)
cylj_t_axis	Vector_3D	(Reference axis t direction)
cylj_rst_coord1	Vector_3D	(Inner surface translational coordinates)
cylj_rst_theta1	Vector_3D	(Inner surface rotational coordinates)
cylj_rst_coord2	Vector_3D	(Outer surface translational coordinates)
cylj_rst_theta2	Vector_3D	(Outer surface rotational coordinates)
cylj_rst_force	Vector_3D	(Translational force vector)
cylj_rst_moment	Vector_3D	(Rotational moment vector)



Warning: The cylindrical joint element has no mass and only increases the stiffness of the nodes in `inner_surface` and `outer_surface`. This additional stiffness is currently not reflected in the computation of the stable time step. A time step scale factor may be required to obtain a stable time step if nodes in the cylindrical joint are part of the element that is controlling the time step. No checks are made of the validity of the stiffness functions provided but the functions should only have values in the upper right and lower left quadrants of a force versus displacement or moment versus rotation plot.

6.2.16 Composite Tet

```

BEGIN TOTAL LAGRANGE SECTION <string>section_name
  FORMULATION = COMPOSITE_TET
  STRAIN INCREMENTATION = <string>
    STRONGLY_OBJECTIVE |
    LOGARITHMIC_MAPPING
    (STRONGLY_OBJECTIVE)
  VOLUME AVERAGE J = <string>ON|OFF (ON)
END [TOTAL LAGRANGE SECTION <string>section_name]

```

The composite tetrahedral element formulation can only be used with 10-noded tetrahedral elements.

As stated in Reference [13], the element is classified as composite because conceptually it consists of 12 sub-elements, each of which is a linear, 4-noded tetrahedron. Thus, the basis functions used to represent nodal fields are piecewise linear within the element. The deformation gradient and stress fields are assumed to live in a linear space, and the resulting gradient operator incorporates

the projection of the piecewise discontinuous gradients into the assumed linear basis. The current implementation does not restrict the mid-edge nodes to be colinear with the vertex nodes, allowing for better geometric resolution for curved bodies.

For hypoelastic constitutive models, the default STRAIN INCREMENTATION option uses a STRONGLY OBJECTIVE algorithm to compute the rate of deformation. The LOGARITHMIC MAPPING option is thought to be more accurate at an additional computational expense.

A volume averaging scheme, applied to the determinant of the deformation gradient and the pressure, is used to avoid volumetric locking. Volumetric locking can occur when the material response involves isochoric motion or nearly incompressible behavior, which are common for ductile materials and rubbery materials respectively. By default the volume averaging scheme is ON.

This is the only option in the TOTAL LAGRANGE SECTION that is production ready. For other information about the TOTAL LAGRANGE SECTION please look at the capabilities in development manual.

6.3 Element-like Functionality

This section describes functionality in Sierra/SM that behaves similarly to finite elements, although are not implemented as such. The functionality described in this section are specified through command blocks that appear in the SIERRA scope and region scope, respectively.

6.3.1 Rigid Body

```
BEGIN RIGID BODY <string>rb_name
  MASS = <real>mass
  POINT MASS = <real>mass [AT <real>X <real>Y <real>Z]
  REFERENCE LOCATION = <real>X <real>Y <real>Z
  INERTIA = <real>Ixx <real>Iyy <real>Izz <real>Ixy
            <real>Iyz <real>Izx
  POINT INERTIA = <real>Ixx <real>Iyy <real>Izz <real>Ixy
                 <real>Iyz <real>Izx
  MAGNITUDE = <real>magnitude_of_velocity
  DIRECTION = <string>direction_definition
  ANGULAR VELOCITY = <real>omega
  CYLINDRICAL AXIS = <string>axis_definition
  INCLUDE NODES IN <string>surface_name
    [IF <string>field_name <|<|=|>|=|> <real>value]
END [RIGID BODY <string>rb_name]
```

A rigid body can consist of any combination of elements—solid elements, structural elements, and point masses—except SPH or peridynamics elements. All nodes associated with a rigid body maintain their relative position to each other as determined at time zero when there is no deformation of the body. This means that the elements associated with the rigid body can translate and rotate through space, but they cannot deform. Element blocks defining the rigid body do not have to be contiguous. They can also adjoin deformable element blocks. Multiple rigid bodies are allowed in a model.



Warning: Truss elements currently do not work correctly when used in a rigid body unless the user includes the `INERTIA` command in the rigid body command block to specify the total mass moment of inertia of the rigid body. The code currently does not compute the inertia about the longitudinal axis of a truss because only the area is input for a truss section. The recommendation is to use a beam section rather than a truss section for rigid bodies.



Warning: The non-deformable nature of the rigid body can put it in conflict with other constraints. Refer to [Appendix D](#) for information on how conflicting constraints are handled.



Warning: Kinematic boundary conditions prescribed on a rigid body must be applied to the rigid body reference location, e.g., its center of mass. Kinematic boundary conditions for a rigid body that are not prescribed on the rigid body reference location will not be enforced. Note that, unlike non-rigid blocks, a rigid body block may rotate about the rigid body reference location if only translational boundary conditions are prescribed, so rotational boundary conditions may also be prescribed to constrain rotational movement. To apply kinematic boundary conditions to a rigid body, use the `BLOCK =` or the `RIGID BODY =` line command with the rigid body's block id or name, respectively, in the appropriate boundary condition command block. Refer to Chapter 7.

Sierra/SM creates a new node for each rigid body in the analysis. The new nodes are true nodes in that they are associated with solution fields such as displacement, velocity, and rotational velocity. These nodes will appear in a results file along with other nodes. The global node number given to the new nodes is simply the total number of nodes in the mesh plus one, repeated for each new rigid body node.

If an element block is declared to be a part of a rigid body, the internal force calculations are not called for the elements in that block and element time steps for explicit dynamics are not computed. Hence, for a model where all the element blocks are included in rigid bodies, there is no estimate for a global explicit dynamics time step. In this case an initial time must be specified in the `TIME CONTROL` command block (Section 3.1.1) with the command line

 **Explicit** `INITIAL TIME STEP = <real>time_step`

For the completely rigid explicit dynamics case, one also should avoid using a value larger than 1.0 for the time step scale factor (see Section 3.1.1). A time step value of 1.0×10^{-6} should work well for most such problems, though any time step may be specified. However since a completely rigid problem does not impose a stability limit on the explicit dynamics time step, specifying too large of a time step could cause loss of accuracy in the solution of the problem (due to numerical integration error, large rotations over a time step, etc.) even though the problem runs to completion. The default value of the time step scale factor is 0.9.

Specification of a rigid body requires the above command block, which appears in the SIERRA scope, plus the `RIGID BODY` command line that appears in the various `SECTION` command blocks described in this chapter. Suppose, for example, `rigidbody_1` consists of element blocks 100, 110, and 280. The `PARAMETERS FOR BLOCK` command blocks for element blocks 100, 110, and 280 must all contain a `SECTION` command line. In each case, the Section must contain a line such as:

```
RIGID BODY = rigidbody_1
```

Once element block(s) have been declared a rigid body and named appropriately (through the Section chosen), that rigid body name must appear as the name in a `RIGID BODY` command block. In the example, a `RIGID BODY` command block with the value for `rb_name` set to `rigidbody_1` must exist, i.e., in the Sierra scope,

```
BEGIN RIGID BODY rigidbody_1
END [RIGID BODY rigidbody_1]
```

The RIGID BODY command block has several optional command lines, composing four groups of commands. One group consists of the MASS, POINT MASS, REFERENCE LOCATION, POINT INERTIA, and INERTIA command lines, a second group consists of the paired MAGNITUDE and DIRECTION command lines, a third group consists of the paired ANGULAR VELOCITY and CYLINDRICAL AXIS command lines, and a final group consists of the INCLUDE NODES IN command line. The command block can include none or any combination of these groups. If none of these commands is included, the command block simply supplies the value for `rb_name`.

Input to the MASS command line consists of a single real number that defines the total mass of the rigid body. If this line command is not present, the mass of the rigid body will be computed using the elements in the rigid body and their densities. This is the translational mass at the reference location. This command does not change the inertia terms.

The POINT MASS command line requires a real number specifying the amount of mass added to the rigid body. Optionally, the location of that mass may be specified with three real numbers. If the location is not given, the additional mass will be placed at the reference location (and will not affect the moments and products of inertia).

The REFERENCE LOCATION command line requires three real numbers defining the reference location for the rigid body. If this line command is not present, the center of mass will be used. The center of mass is calculated from the mesh and the element densities.

Input to the INERTIA command line consists of six real numbers. If present, this command line will set the inertia for the rigid body. If it is not present, moments and products of inertia are computed for the rigid body based on the reference location of the rigid body and the element masses.

Input to the POINT INERTIA command line consists of six real numbers that define moments (I_{xx} , I_{yy} , I_{zz}) and products (I_{xy} , I_{yx} , I_{zx}) of inertia to be added to the inertia tensor of the rigid body. This modified inertia tensor (rather than the inertia tensor based solely on element mass) is then used to calculate the motion of the rigid body.

If a rigid body contains an element that has rotational resistance (e.g., shell, beam, etc.) the nodal moment of inertia of that element is added to the diagonal components of the rigid body's inertia tensor.

An initial, translational-only velocity for the rigid body at the reference location should be specified with the MAGNITUDE and DIRECTION command lines. The MAGNITUDE command line gives the magnitude of the initial velocity applied at the reference location, and the DIRECTION command line gives a defined direction.

An initial rotational and translational velocity for a rigid body can be specified with the ANGULAR VELOCITY and CYLINDRICAL AXIS command lines. The ANGULAR VELOCITY command line gives the initial translational velocity of and rotational velocity about the reference location due to an angular velocity about some defined axis given on the CYLINDRICAL AXIS command line. If the defined cylindrical axis passes through the reference location, this command will impart only an initial rotational velocity to the rigid body. If the axis does not pass through the reference

location, this command will impart both an initial translational velocity and the initial rotational velocity to the rigid body.

The `INCLUDE NODES IN` command line allows a rigid body to include nodes of a surface or block of the mesh. Optionally, the nodes in the surface or block will be included in the rigid body only if the value of a field on the nodes meets a given criterion. For example, consider the rigid body block below.

```
begin rigid body rigid1
  include nodes in block_1
  include nodes in surface_3 if height = -1.0
end
```

Rigid body `rigid1` will include all nodes in `block_1` and those nodes on `surface_3` whose value of the `height` field are equal to `-1.0`. The optional field (`height` in this case) must be a field known in the analysis. This field may be read in from restart, be a native field initialized upon start up, or be an initialized user-defined field. Since the entire set of nodes in `block_1` will be in the rigid body, internal forces and critical time steps (for explicit) will not be computed for elements in `block_1`.

In summary, if a rigid body is used in an analysis, one or more of the following steps must be carried out:

- Include a `RIGID BODY` command block in the `SIERRA` scope. If desired, set reference location, mass, point mass, etc.
- Create a rigid body using one or more element blocks (except `PARTICLE` or `PERIDYNAMICS` element blocks). A `RIGID BODY` command line must appear in the `SECTION` command block used in the `PARAMETERS FOR BLOCK` command block for any element block associated with a rigid body.
- Include point mass element blocks with the rigid body if appropriate. To include point mass element blocks in a rigid body, a `RIGID BODY` command line must appear in the `SECTION` command block used in the `PARAMETERS FOR BLOCK` command block for those point mass element blocks.
- Associate an initial velocity or initial rotation about an axis with the rigid body, if appropriate. If any of the blocks associated with a rigid body have been given an initial velocity or initial rotation, the rigid body must have the same specification for the initial velocity or initial rotation.

The above steps involve many different command blocks. To demonstrate how to fully implement a rigid body, a specific example that exercises the various options available to a user is provided.

Suppose it is desired to create a rigid body named `part_a` consisting of three element blocks. Two of the element blocks, element block 100 and element block 535, are eight-node hexahedra; one of the element blocks, element block 600, consists of only point masses. The `RIGID BODY` command block, `SECTION` command block, and the element blocks to be associated with the rigid body are as follows:

```
begin solid section hex_section
```

```

    rigid body = part_a
end
begin point mass section pm_section
    rigid body = part_a
    volume = 0.1
end

begin parameters for block block_100
    material = steel
    model = elastic
    section = hex_section
end
begin parameters for block block_535
    material = aluminum
    model = elastic
    section = hex_section
end
begin parameters for block block_600
    material = mass_for_pointmass
    model = elastic
    section = pm_section
end

```

Suppose it is desired to have the rigid body initially rotating at 600 radians/sec about an axis parallel to the x -axis and passing through a point at coordinates (0, 10, 20). This axis would be defined using the following set of command lines:

```

define direction parallel_to_x with vector 2.0 0.0 0.0
define point off_axis with coordinates 0.0 10.0 20.0
define axis body_axis with point off_axis direction parallel_to_x

```

The initial angular velocity specification in the RIGID BODY command block is as follows:

```

begin rigid body part_a
    cylindrical axis = body_axis
    angular velocity = 600
end rigid body part_a

```

6.3.1.1 Multiple Rigid Bodies from a Single Block

Typically, all the elements in a block are assigned to a single rigid body. However, it is sometimes necessary to define a large number of rigid bodies. It would be unwieldy to create a separate block for each rigid body. To avoid this, it is possible to create an element attribute in the input mesh file that contains an integer ID used to denote the rigid body in which each element should belong. The RIGID BODIES FROM ATTRIBUTES command is used to associate the attribute with the rigid body IDs. This command must be used in conjunction with the RIGID BODY command.

These commands are used in the context of the section block. They are shown here in the `SOLID SECTION` block, but they can be used with any section type that supports rigid bodies.

```
BEGIN SOLID SECTION <string>sec_name
  RIGID BODIES FROM ATTRIBUTES = <integer>first_id to
    <integer>last_id
  RIGID BODY = <string>rigid_body_name
END
```

If the `RIGID BODIES FROM ATTRIBUTES` command is used, a series of rigid bodies will be created. These rigid bodies are named using the specified `rigid_body_name`, followed by an underscore (`_`), and then by an integer ID, which ranges between the specified value of `first_id` and `last_id`. Elements having an attribute value equal to a given ID are added to the corresponding rigid body.

When referencing the rigid body name for to apply boundary conditions, it is important to specify the entire name of the rigid body, including the underscore and ID. For example, a name such as `part_a_1` would be used in a boundary condition. Similarly, specific rigid bodies created with `RIGID BODIES FROM ATTRIBUTES` can be have rigid body properties `REFERENCE LOCATION`, `MASS`, `POINT MASS`, `INERTIA`, and `POINT INERTIA` set using a name such as `part_a_1`.

6.3.2 Output

Sierra/SM automatically outputs quantities such as displacement for the reference location of the rigid body. The name assigned to a rigid body will be used to construct variable names that give the quantities. This lets the user identify the output associated with a rigid body based on the name assigned to the rigid body.

Immediately before the results file is written, the accelerations for nodes associated with a rigid body are updated to reflect the accelerations due to the rigid-body constraints. This ensures that the accelerations sent to the results output are correct for a given time.

Sierra/SM automatically generates and outputs global data associated with the rigid body (e.g., displacement and quaternion). See Section 9.9 for details on global variable output. Individual field components may be output as discussed in Section 9.1.4 and Table 9.3. In general the field component `fieldi` of the rigid body named `part_a` will be written to the output file(s) with the name `FIELDI_PART_A`.

In addition to being represented in the mesh file by the grouping of element objects that represent the rigid body each rigid body will also be represented by a sphere element. These rigid body elements will be placed in a new element block in the results output file named “RigidBodyOutput-Part”. Net quantities of the rigid body such as rotation rate, net force, and velocity can be plotted on these sphere elements.

6.4 Mass Property Calculations

```
BEGIN MASS PROPERTIES
#
# block set commands
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
#
# structure command
STRUCTURE NAME = <string>structure_name
END [MASS PROPERTIES]
```

Sierra/SM automatically gives mass property information for the total model, which consists of all the element blocks. (The mass for the total model, for example, is the total mass of all the element blocks.) Sierra/SM also automatically outputs mass property information for each element block.

In addition to the generated mass property information, Sierra/SM allows the user to optionally define a structure composed of multiple element blocks and calculate the mass properties for this structure using the `MASS PROPERTIES` command block in the region scope.

For the total model, each element block, and any user-defined structure, Sierra/SM reports the mass and the center of mass in the global coordinate system. It also reports the moments and products of inertia, as computed in the global coordinate system, about the center of mass.

The `MASS PROPERTIES` command block contains two groups of commands—block set and structure. Each of these groups is independent of the other. Following are descriptions of the two command groups.

6.4.1 Block Set Commands

The `block set commands` portion of the `MASS PROPERTIES` command block defines a set of blocks for which mass properties are being requested, and can include some combination of the following command lines:

```
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of blocks. See [Section 7.1.1](#) for more information about the use of these command lines for creating a set of blocks used by the command block. There must be at least one `BLOCK` or `INCLUDE ALL BLOCKS` command line in the command block.

The `REMOVE BLOCK` command line allows the user to delete blocks from the set specified in the `BLOCK` and/or `INCLUDE ALL BLOCKS` command line(s) through the string list `block_names`. The `REMOVE BLOCK` command line is typically used with the `INCLUDE ALL BLOCKS` command line. To include all but a few of the element blocks, a combination of the `REMOVE BLOCK` command line and `INCLUDE ALL BLOCKS` should minimize input information.

Suppose that only one element block, `block_300`, is specified on the `BLOCK` command line. Then only the mass properties for that block will be calculated. If several element blocks are specified on the `BLOCK` command line, then that collection of blocks will be treated as one entity, and the mass properties for that single entity will be calculated. For example, if two element blocks, `block_150` and `block_210`, are specified on the `BLOCK` command line, the total mass for the two element blocks will be reported as the total mass property.

6.4.2 Structure Command

The output for the mass properties will be identified by the command line:

```
STRUCTURE NAME = <string>structure_name
```

where the string `structure_name` is a user-defined name for the structure.

6.5 Element Death

```

BEGIN ELEMENT DEATH <string>death_name
#
# block set commands
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
#
# criterion commands
CRITERION IS AVG|MAX|MIN NODAL VALUE OF
    <string>var_name <|<=>|>=>|> <real>tolerance
CRITERION IS ELEMENT VALUE OF
    <string>var_name <|<=>|>=>|> <real>tolerance [KILL WHEN
    <integer>num_intg INTEGRATION POINTS REMAIN]
CRITERION IS GLOBAL VALUE OF
    <string>var_name <|<=>|>=>|> <real>tolerance
CRITERION IS ALWAYS TRUE
MATERIAL CRITERION = ON|OFF(OFF) [KILL WHEN
    <integer>num_intg INTEGRATION POINTS REMAIN]
#
# subroutine commands
ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#
# evaluation commands
CHECK STEP INTERVAL = <integer>num_steps
CHECK TIME INTERVAL = <real>delta_t
DEATH START TIME = <real>time
#
# miscellaneous option commands
SUMMARY OUTPUT STEP INTERVAL = <integer>output_step_interval
SUMMARY OUTPUT TIME INTERVAL = <real>output_time_interval
DEATH ON INVERSION = OFF|ON(OFF)
DEATH ON ILL DEFINED CONTACT = OFF|ON(OFF)
DEATH STEPS = <integer>death_steps(1)
FORCE VALID ACME CONNECTIVITY
AGGRESSIVE CONTACT CLEANUP = <string>OFF|ON(OFF)
DEATH METHOD = <string>DEACTIVATE ELEMENT|DEACTIVATE NODAL MPCs|
DISCONNECT ELEMENT|(DEACTIVATE ELEMENT)
#
# death on proximity commands

```

Explicit

```

BEGIN DEATH PROXIMITY
  BLOCK = <string list>block_names
  NODE SET = <string list>nodelist_names
  SURFACE = <string list>surface_names
  REMOVE BLOCK = <string list>block_names
  REMOVE NODE SET = <string list>nodelist_names
  REMOVE SURFACE = <string list>surface_names
  TOLERANCE = <real>search_tolerance
END [DEATH PROXIMITY]
#

```

Explicit

```

# particle conversion commands
BEGIN PARTICLE CONVERSION
  PARTICLE SECTION = <string>section_name
  NEW MATERIAL = <string>material_name
  MODEL NAME = <string>model_name
  TRANSFER = <string>NONE|ALL (ALL)
  MAX NUM PARTICLES = <integer>num(1)
  MIN PARTICLE CONVERSION VOLUME = <real>vol
END [PARTICLE CONVERSION]
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
REMOVE OVERLAP AFTER DEATH = ON|OFF (ON)
#
# cohesive zone setup commands
COHESIVE SECTION = <string>sect_name
COHESIVE MATERIAL = <string>mat_name
COHESIVE MODEL = <string>model_name
COHESIVE ZONE INITIALIZATION METHOD = <string>NONE|
  ELEMENT STRESS AVG(NONE)
ALWAYS EVALUATE ALL CRITERION = OFF|ON (OFF)
END [ELEMENT DEATH <string>death_name]

```

The `ELEMENT DEATH` command block is used to remove elements from an analysis. For example, the command block can be used to remove elements that have fractured, that are no longer important to the analysis results, or that are nearing inversion (explicit only). This command block is located within the region scope. The name of the command block, `death_name`, is user-defined and can be referenced in other commands to update boundary or contact conditions based on the death of elements creating new exposed surfaces.

Any element in an element block or element blocks selected in the `ELEMENT DEATH` command block is removed (killed) when one of the criteria specified in the `ELEMENT DEATH` command block is satisfied by that element. When an element dies, it is removed permanently. Any number of `ELEMENT DEATH` command blocks may exist within a region.

When an element is killed, the contribution of that element's mass to the attached nodal mass is removed from the attached nodes. If all the elements attached to a node are killed, the mass for the node and all associated nodal quantities will be set to zero. If all the elements in an explicit region are killed, the analysis will terminate.



Known Issue: Termination when all elements are killed is not yet implemented for implicit element death.

In implicit calculations, elements may be killed based off all available criterion and used in conjunction with control failure in a multilevel solver block. However, material criterion is the only criterion that is truly integrated with the control failure to control the failure progression. Control failure is described in Section 4.7.

In explicit calculations, elements may be killed based on any internal variable, derived variable, material state variable, or user defined variable. Note that user-defined variables for triggering element death should be used with care. There are timing and parallel issues regarding the use of a user-defined variable with element death. User subroutines may work for element death in some situations where one might want to reference a user variable.

The quantity used in an element death criterion statement should have only a single value at each integration point. For example, the following command:



Explicit Only



Explicit

```
criterion is element value of stress(xx) > 10.0
```

is unambiguous and valid. However this command:



Explicit

```
criterion is element value of stress > 10.0
```

is ambiguous, and will result in an error. Stress is a 3x3 tensor and there is no rational way to compare that tensor to the scalar value 10. If a variable is defined on multiple integration points, the criterion will be triggered if the value is exceeded at any of the integration points. If the command:



Explicit

```
criterion is element value of stress(xx,:) > 10.0
```

were used with shells with five integration points, the shells would be killed if the value of `stress(xx)` at any of the five integration points exceeded 10.

The `ELEMENT DEATH` command block contains five groups of commands—block set, criteria, evaluation, miscellaneous, and cohesive zone setup. The command block must contain commands from the block set and criteria groups. Command lines from the evaluation and miscellaneous groups are optional, as are the cohesive zone commands.

Following are descriptions of the different command groups, an example of using the `ELEMENT DEATH` command block, and some concluding remarks related to element death visualization.

6.5.1 Block Set Commands

```
BLOCK = <string list>block_names  
INCLUDE ALL BLOCKS  
REMOVE BLOCK = <string list>block_names
```


The `block set` commands portion of the `ELEMENT DEATH` command block defines a set of blocks for selecting the elements to be referenced. These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of blocks, as described in Section 7.1.1.

Element death must apply to a group of elements. There are two commands for selecting the elements to be referenced: `BLOCK` and `INCLUDE ALL BLOCKS`. In the `BLOCK` command line, the user may list a series of blocks through the string list `block_names`. This command line may also be repeated multiple times. The `INCLUDE ALL BLOCKS` command line adds all the element blocks present in the region to the current element death definition. There must be at least one `BLOCK` or `INCLUDE ALL BLOCKS` command line in the `ELEMENT DEATH` command block.

The `REMOVE BLOCK` command line allows the user to delete blocks from the set specified in the `BLOCK` and/or `INCLUDE ALL BLOCKS` command line(s) through the string list `block_names`. The `REMOVE BLOCK` command line is typically used with the `INCLUDE ALL BLOCKS` command line. To include all but a few of the element blocks, a combination of the `REMOVE BLOCK` command line and `INCLUDE ALL BLOCKS` command line should minimize input information.

6.5.2 Criterion Commands

Any combination of death criteria (`CRITERION IS NODAL`, `CRITERION IS ELEMENT`, `CRITERION IS GLOBAL`, `CRITERION IS ALWAYS TRUE`, `ELEMENT BLOCK SUBROUTINE`, `MATERIAL CRITERION` as appropriate in explicit or implicit) can be specified within a single `ELEMENT DEATH` command block. However, only one user subroutine criterion (available for explicit only) may appear in a set of criteria command lines. If multiple death criteria are specified for a given element, it will be killed when the first of those criteria is met. In other words, element death with multiple criteria is an `OR` condition rather than an `AND` condition.

An explicit dynamics simulation with a material that has both a tension cutoff and a compression cutoff is an example for which element death with multiple criteria would be used—one criterion to set failure in tension and another criterion to set failure in compression. If either the tension cutoff value, as set by the tension criterion, or the compression cutoff value, as set by the compression criterion, was exceeded for an element, the element would be killed.

As a second explicit dynamics example, consider a problem using a nodal criterion and a user subroutine criterion. For this second example, the user is precluded from using a second subroutine criterion because one already exists. A combination of nodal, element, global, or material criteria could be added to the existing nodal and subroutine criteria, but another subroutine criterion could not be added.



6.5.2.1 Nodal Variable Death Criterion

```
CRITERION IS AVG|MAX|MIN NODAL VALUE OF
<string>var_name <|<=|=|>=|> <real>tolerance
```

Any nodal variable may be used by an element death criterion. The input parameters are described as follows:

- Nodal variables are present on the nodes of an element, and these nodal values must be reduced to a single element value for use by the criterion. The available types of reduction



are AVG, which takes the average of the nodal values; MAX, which takes the maximum of the nodal values; and MIN, which takes the minimum of the nodal values.

- The string `var_name` gives the name of the variable. See Section 9.10 for a listing of the variables. Parenthesis syntax may be used in the variable name for selection specific variable components or integration points, see Section 9.1.
- The specified variable, with an optional component specification if the variable has components, may be compared to a given tolerance with one of five operators. The operator is specified with the appropriate symbol for less than (<) less than or equal to (<=), equal to (=), greater than or equal to (>=), or greater than (>) The given tolerance is specified with the real value `tolerance`.

6.5.2.2 Element Variable Death Criterion

```
CRITERION IS ELEMENT VALUE OF
<string>var_name[(<integer>component_num)]
<|<=|=|>=> <real>tolerance |
<string>derived_quantity[(<integer>int_num)]
<|<=|=|>=> <real>tolerance [KILL WHEN <integer>num_intg
INTEGRATION POINTS REMAIN]
```

Any element variable may be used by an element death criterion. An element variable is present on the element itself, so no reduction is required, which is why the first line in the format of the above command line differs from that of the nodal criterion command line in Section 6.5.2.1.

For a criterion using an element variable, the variable name, component or integration point number, and tolerance can be specified in the same manner as defined for the nodal criterion command line.

Element state variables can also be used as criteria for element death. The syntax used to specify the variable name differs if a specific integration point is desired.

- For LAME solid material models, the variable name is called out directly by name.
- For shell material models, the variable name is called out directly by name. By default the element will be killed if the criteria is met at any integration point. Parenthesis syntax (See Section 9.1) may be used to kill based on the state of a specified integration point. For example:

```
criterion = element value of crack_opening_strain(2) > 0.01
```

would use the value of the state variable `crack_opening_strain` at the second shell integration point through the shell thickness as the criterion for element death.

Refer to Section 9.10.2 for tables with listings of state variables for the various material models.

Derived quantities can also be used for element death. For example the following command:

```
criterion = element value of von_mises > 1.0e+05
```

will cause elements to be killed any of the integration points reach a critical value of `von_mises`. Alternatively, the command:

```
criterion = element value of von_mises(3) > 1.0e+05
```

is used to kill an element if a specific integration point reaches the critical value of `von_mises`.

For elements with multiple integration points, if an element criterion is used with an integration point variable, the integration points are killed individually as the criterion is exceeded on each integration point. The integration points are killed by linearly tapering down all components of the stress over the specified number of death steps. See Section 6.5.4.4 for more information on death steps.

The optional `KILL WHEN num_intg INTEGRATION POINTS REMAIN` portion of the command line is used to kill the entire element when enough integration points have been killed. This occurs when the number of remaining points drops down to the specified `num_intg`. For example, for shells with five integration points, the following command:

```
criterion is element value of stress(xx,:) > 10.0 kill when 2 \cmd{\#}  
integration points remain
```

will check the `xx` component of `stress` at each integration point. When the first three integration points reach that criterion, their stresses are linearly tapered down to zero over the specified number of death steps. Once the contributions from each of those three integration points have completely decayed to zero, leaving two remaining integration points, the element is killed. By default the element itself is killed after the first integration point dies.



6.5.2.3 Global Death Criterion

```
CRITERION IS GLOBAL VALUE OF  
<string>var_name[(<integer>component_num)]  
<|<=|=|>=>|> <real>tolerance
```

Any global variable may be used in an element death criterion. Once the global criterion is reached, all elements specified in the `ELEMENT DEATH` command block are killed. The variable name, component number, and tolerance can be specified in the same manner as defined for the nodal or element criterion command line.

The input parameters are described as follows:

- The string `var_name` gives the name of the global variable. See Section 9.10 for a listing of the global variables.
- Parenthesis syntax may be used in the variable name to specify specific components of the variable. See Section 9.1 for more information.
- The specified variable, with an optional component specification if the variable has components, may be compared to a given tolerance with one of five comparison operators as follows:
 - less than (`<`),
 - less than or equal to (`<=`),
 - equal to (`=`),
 - greater than or equal to (`>=`), or
 - greater than (`>`).

The given tolerance is specified with the real value `tolerance`.



6.5.2.4 Always Death Criterion

CRITERION IS ALWAYS TRUE

This command forces the criterion that determines whether to kill elements specified in the current death block to always be true. In other words, it causes all of those elements to immediately be killed. This command can be used to instantaneously kill a set of elements or convert that set of elements to particles. Converting a set of elements to particles in this manner at the beginning of an analysis can be an alternative to generating a mesh with particle elements.

This command is often used together with the `DEATH START TIME` command (see Section 6.5.3) to cause a set of elements to be killed or converted to particles at a certain time. This can be used as an alternative to block deactivation using the `ACTIVE FOR PROCEDURE` command (see Section 6.1.6.9).

6.5.2.5 Material Death Criterion

MATERIAL CRITERION = ON|OFF (OFF) [KILL WHEN
<integer>num_intg INTEGRATION POINTS REMAIN]

Some material models have a failure criterion. When this material failure criterion is satisfied within an element the material models reduce the stress in these fractured or disintegrated elements to zero. The `MATERIAL CRITERION` command line can be used to link the decay of stress to zero with deletion of the element from an analysis. Removal of the fractured elements can speed computations, enhance visualization, and prevent spurious inversion of these elements that may stop the analysis.

The material models that currently support the use of the `MATERIAL CRITERION` command line are:

- `ELASTIC_FRACTURE` (Solid only, see Section 5.2.4)
- `DUCTILE_FRACTURE` (Solid only, see Section 5.2.7)
- `ML_EP_FAIL` (Solid and Shell, see Section 5.2.9)
- `PLANE_STRESS_RATE_PLASTICITY` (Shell only, see Section 5.2.32)

These material models compute the element variables “failure_flag”, “failure_decay”, and “failure_measure”. In its normal computations the “failure_flag” will have a value of 0.0. When the material is decaying the “failure_flag” will have a value of 3.0 and when it is finished decaying the “failure_flag” will have a value of 4.0. As the material is decaying the “failure_decay” will reduce from 1.0 to 0.0. The “failure_measure” value and function is explained in the Control Failure Section 4.7.

Element death will kill an element based on a material criterion when the material model has set the “failure_flag” to 4.0. For a single integration point element, this occurs when the one integration point in the element reaches a “failure_flag” = 4.0. For elements with multiple integration points, the default behavior is for the element to be killed when all but one of the integration points has failed. This behavior is the default because for multi-integration point elements, particularly shells, if there is only a single integration point left, the element is severely under-integrated. The final

integration point will generally not attract more load and will never fail. This default behavior can be changed by using the optional `KILL WHEN num_intg INTEGRATION POINTS REMAIN` command. In this command, `num_intg` specifies the number of remaining integration points when the element is to be killed. By default the entire element dies if more than half of the integration points die.

Consider an element block named `part1_ss304` that references a material named `SS304`. This material, `SS304`, uses the `DUCTILE_FRACTURE` material model (see Section 5.2.7). Consider a second element block named `ring5_al6061` that references a material named `al6061`. This material uses the `ML_EP_FAIL` material model (see Section 5.2.9). An `ELEMENT DEATH` command block with the command lines

```
BLOCK = part1_ss304 ring5_al6061
MATERIAL CRITERION = ON
```

would signal any element in `part1_ss304` that fails according to the material model `DUCTILE_FRACTURE` (in material `SS304`) and any element in `ring5_al6061` that fails according to the material model `ML_EP_FAIL` (in material `al6061`) to be killed by element death.



6.5.2.6 Subroutine Death Criterion

```
ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

A death criterion can be specified via a user-defined subroutine (see Chapter 11), which is invoked by the `ELEMENT BLOCK SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user. The user-defined subroutine for element death must be an element subroutine signature (see Section 11.2.2). The element subroutine will return an output values array and a flag array of one flag per element (see Table 11.2 in Chapter 11). The output values array is ignored. Death is determined by the flag return value. For user-defined subroutines, a flag return value of `-1` indicates that the element should die. A flag return value greater than or equal to `0` indicates that the element should remain alive.

Following the `ELEMENT BLOCK SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Section 11.

6.5.3 Evaluation Commands

```
CHECK STEP INTERVAL = <integer>num_steps
```

```
CHECK TIME INTERVAL = <real>delta_t
DEATH START TIME = <real>time
```

Evaluation of element death criteria may be time consuming. Additionally, reconstruction of contact or other boundary conditions after element death can be time consuming. For these reasons, three command lines are available for determining the frequency at which element death is evaluated. The default is to evaluate element death at every time step. The number of times at which element death evaluation is done may be limited using the following commands.

- The `CHECK STEP INTERVAL` command line instructs element death to evaluate the element death criteria only every `num_steps` time steps.
- The `CHECK TIME INTERVAL` command line instructs element death to evaluate the element death criteria only every `delta_t` time units.
- The `DEATH START TIME` command line instructs element death not to evaluate death criteria before a user-specified time, as given by the real value `time`.

Both the `CHECK STEP INTERVAL` and `CHECK TIME INTERVAL` command lines may be used in a command block. Evaluations for element death will be made at both the time and step intervals if both of these command lines are included.

All three of `CHECK STEP INTERVAL`, `CHECK TIME INTERVAL`, and `DEATH START TIME` are optional command lines.

6.5.4 Miscellaneous Option Commands

The command lines listed below need not be present in the `ELEMENT DEATH` command block unless the conditions addressed by each call for their inclusion.

6.5.4.1 Summary Output Commands

At the end of a run, a summary of all element death blocks is output to the log file. The `SUMMARY OUTPUT STEP INTERVAL` or `SUMMARY OUTPUT STEP INTERVAL` commands can be used to request that the summary be output to the log file at regular intervals during the run. The `SUMMARY OUTPUT TIME INTERVAL` command results in the summary being printed every `output_step_interval` steps, while the `SUMMARY OUTPUT TIME INTERVAL` results in the summary being printed once every `output_time_interval` time units. These two commands can be supplied individually, together or not at all. If neither are used, the summary is printed only at the end of execution.

This command applies to all element death blocks. If these commands appear in multiple element death blocks, the values specified in the last instance of each of these commands prevails.

6.5.4.2 Death on Inversion

```
DEATH ON INVERSION = ON|OFF (OFF)
```

If the `DEATH ON INVERSION` command line is set to `ON`, any element that inverts will be killed. This command currently only works for uniform gradient eight-noded hexahedral elements, four-, eight-, and ten-noded tetrahedral elements, and four-noded shells.



6.5.4.3 Death on Ill-defined Contact

```
DEATH ON ILL DEFINED CONTACT = ON|OFF(OFF)
```

If the `DEATH ON ILL DEFINED CONTACT` command line is set to `ON`, any element that would create invalid geometry for contact will be killed. This currently only works for uniform gradient eight-noded hexahedral elements and four-noded tetrahedra elements. An example of an element that creates an invalid contact geometry is an element that is partially inverted in such a way that some regions of the element have negative volume and exterior faces of the element point in the wrong directions.

A highly deformed element can have a positive total volume, permitting internal force to still be calculated, but have inverted local regions that make it invalid for contact. For this reason, killing elements due to ill-defined contact can be important to obtain solutions for contact problems in which elements are experiencing large distortions. Using death on inversion alone is not sufficient for those cases.

6.5.4.4 Death Steps

```
DEATH STEPS = <integer>death_steps(1)
```

If the `DEATH STEPS` command line is used and the value for `death_steps` is set to a value greater than 1, all components of the stress and the hourglass forces in a killed element are linearly scaled down until they reach 0 over the specified number of steps. Gradually releasing the energy due to killing an element over a number of steps can avoid instabilities that might occur if that element were suddenly killed. If this is done over an excessively large number of steps, however, the killed element may participate in the analysis long after it is relevant, and may cause problems if it undergoes excessive deformation or inverts while its stresses are being tapered off. The default number of steps, as provided in the integer value `death_steps`, is 1.

The value selected for `death_steps` will depend on the loads, material models, and model complexity associated with an analysis. A small number (3 to 5) may be sufficient to prevent instabilities for most cases. However, it may be necessary to use a value of 10 or larger.

If an element death criterion is used in an element with multiple integration points, the stresses for the integration points that have exceeded the criterion are decayed individually over the specified number of death steps. When the required number of death steps for an integration point has been exceeded, that integration point no longer contributes to the element internal force, and is considered dead. When the required number of integration points to kill the entire element (based on the user-input `KILL WHEN num_intg INTEGRATION POINTS REMAIN` – See Section 6.5.2.2) have reached that state, the process of killing the element is initiated, and the stresses in the remaining integration points and the hourglass forces are all scaled down over the specified number of death steps.

If an element reaches a death criteria and while in the process of fading out reaches another death criteria whichever death criteria has the lowest number of remaining steps will be used. For example if a material has a death criteria on stress and fades out of 10 death steps and in the process



of that fade out has a bad shape and is killed in a single death step the single death step inversion criteria will overwrite the stress fade out criteria.

6.5.4.5 Degenerate Mesh Repair

`FORCE VALID ACME CONNECTIVITY`

If the `FORCE VALID ACME CONNECTIVITY` command line is present, degenerate mesh occurrences will be repaired. Element death has the possibility of creating degenerate mesh occurrences that will not be accepted by the ACME (see Reference [14]) contact algorithms used by Presto. For example, if two continuum elements are connected only by an edge, ACME will not accept the mesh as a valid mesh. This degenerate mesh occurrence (continuum elements connected only at an edge) is repaired by deleting all elements attached to the offending edge, if this repair option is enabled.

See Section 8.4.6 for more information on how contact interacts with element death.

If a `CONTACT DEFINITION` command block is not present, but it is desired to repair degenerate mesh occurrences, the `FORCE VALID ACME CONNECTIVITY` command line should be included.

6.5.4.6 Aggressive Contact Cleanup

`AGGRESSIVE CONTACT CLEANUP = <string>OFF|ON(OFF)`

The `AGGRESSIVE CONTACT CLEANUP` command line enables the use of element death in an attempt to guarantee a valid contact mesh. Certain complex meshes with extensive element death can be problematic for the contact algorithms on some mesh decompositions. This command allows element death to query the contact algorithm and then remove any elements that the contact algorithm flags as having the potential to cause contact to fail.

6.5.4.7 Death Method

`DEATH METHOD = <string>DEACTIVATE ELEMENT|DEACTIVATE NODAL MPCs|
DISCONNECT ELEMENT|(DEACTIVATE ELEMENT)`

The `DEATH METHOD` command specifies what happens when an element meets the death criterion. The following strings can be used as arguments to this command: `DEACTIVATE ELEMENT`, `DEACTIVATE NODAL MPCs`, `DISCONNECT ELEMENT`. The behavior controlled by these options is described below.

- With the default option, `DEACTIVATE ELEMENT`, the element is deactivated, effectively removing it from the mesh.
- The `DEACTIVATE NODAL MPCs` option removes the nodes of a killed element from any multi-point constraints. This only has an effect if the element has nodes that are in multi-point constraints. The multi-point constraint deactivation option can be used to break an element away from the mesh, allowing it to move independently. It can also be used to activate cohesive zones.

- The `DISCONNECT ELEMENT` option disconnects the element from the mesh, allowing the element to move independently. The disconnected element will no longer share any nodes with neighboring elements, and will only interact with the remainder of the mesh through contact.
- The `INSERT COHESIVE ZONE` option has been deprecated.

When using element disconnection, it is necessary to use at least some hourglass stiffness and/or viscosity to prevent large deformations of the disconnected elements in zero energy modes. In addition, it is recommended that a secondary shape-based criterion be used in a separate element death command block to fully remove the disconnected elements if they begin to deform too much. The following example of an additional element death block to be used together with element disconnection shows recommended shape criteria:

```
begin element death
  include all blocks
  #Kill an element if it has negative volume
  death on inversion = on
  #Kill an element if it has become locally inverted (i.e., concave)
  criterion is element value of nodal_jacobian_ratio <= 0.0
  #Kill an element that is becoming very large (shells only)
  criterion is element value of perimeter_ratio > 10.0
end
```



6.5.4.8 Death on Proximity

```
BEGIN DEATH PROXIMITY
  BLOCK = <string list>block_names
  NODE SET = <string list>nodelist_names
  SURFACE = <string list>surface_names
  REMOVE BLOCK = <string list>block_names
  REMOVE NODE SET = <string list>nodelist_names
  REMOVE SURFACE = <string list>surface_names
  TOLERANCE = <real>search_tolerance
END [DEATH PROXIMITY]
```

Sierra/SM provides the option to kill elements that get within a search tolerance of nodes in a specified mesh entity set. Proximity is measured by the smallest distance from the nodes and element centroids of the elements specified in the `BEGIN ELEMENT DEATH` command block to the nodes in the mesh entity set defined in the command block `BEGIN DEATH PROXIMITY`.

The commands `BLOCK`, `NODE SET`, `SURFACE`, `REMOVE BLOCK`, `REMOVE NODE SET`, and `REMOVE SURFACE` inside the command block `BEGIN DEATH PROXIMITY` form the mesh entity set of nodes used to search against for the proximity measure.

The command `TOLERANCE` specifies the distance tolerance. Once an element's proximity measure is less than or equal to the distance tolerance the element will be killed.



6.5.4.9 Particle Conversion

```
BEGIN PARTICLE CONVERSION
  PARTICLE SECTION = <string>section_name
  NEW MATERIAL = <string>material_name
    MODEL NAME = <string>model_name
  TRANSFER = <string>NONE|ALL (ALL)
  MAX NUM PARTICLES = <integer>num(1)
  MIN PARTICLE CONVERSION VOLUME = <real>vol
END [PARTICLE CONVERSION]
```

Sierra/SM provides the option to convert elements to particles when they are killed, rather than simply removing them from the mesh. This option is activated by using the `BEGIN PARTICLE CONVERSION` command block inside the `ELEMENT DEATH` command block. Particle conversion can be useful when elements become too highly deformed to be represented by solid elements. This allows for the highly deformed solid elements to be removed from the mesh, while retaining the mass associated with those elements in the form of particles.

The name of a particle section block (defined with a `PARTICLE SECTION` command block, see Section 6.2.12) must be provided with the `PARTICLE SECTION` command line to define the properties of the created particles. Particles may be created that use the SPH or mass particle formulation, as defined by the section.

The material models for the created particles may be optionally prescribed by providing the name and model for a defined material in the `NEW MATERIAL` command. By default, the particle uses the material model of the solid element from which it has been converted, and all element data that can be retained, including material state variables, are transferred during conversion. This behavior may be modified by specifying the transfer type through the `TRANSFER` command. If all the applicable element data should be transferred, the `ALL` option should be specified (this is the default if no transfer type is specified). If the particles should be created with reinitialized state data, the `NONE` option should be specified. Moreover, if a new material type is specified for conversion, the `TRANSFER` option will apply, but likely much of the material state data will not be transferable and will be lost as the element is converted to a particle. The best option when using a new material is to also specify a transfer type of `NONE`.

Particle conversion works with most element types including hexahedra, tetrahedra, shells, beams, and other particles. The converted particle is placed at the centroid of the removed element. For shells, beams and particles the new particle is placed inside of the lofted volume that the structural element represents. The mass of the particle equals the mass of the converted element. The velocity and/or temperature of the converted particle is set to the average velocity/temperature of the source element's nodes. The radius of the converted particle is computed to allow the particle to fit inside the void space left by removal of the solid element. If no particle can fit in that space, no particle is created. This typically occurs when an element has inverted or the center of mass is not on the interior of the element, as seen in Figure 6.27.

A single element can optionally be converted into multiple particles. The number of particles to convert to is specified with the `MAX NUM PARTICLES` command and `MIN PARTICLE CONVERSION VOLUME` command. The usage of these commands is as follows:

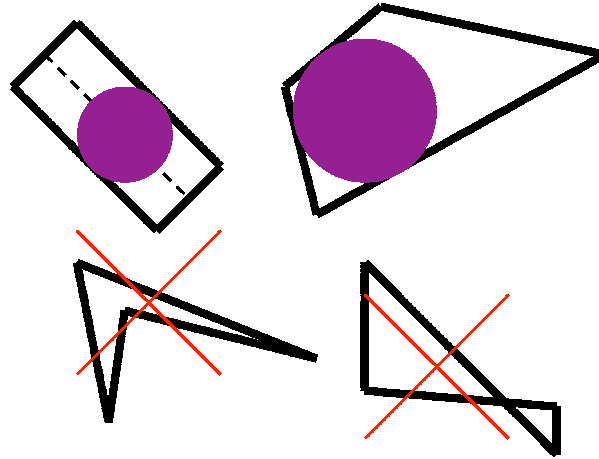


Figure 6.27: Examples of how an element is converted and when it is not converted.

If neither of these command lines is present, one particle is placed at the centroid of the element.

If only the `MAX NUM PARTICLES` command line is present, the element will be cut into the specified number of roughly equal volume sections. A particle will then be placed at the centroid of each of these sections.

If only the `MIN PARTICLE CONVERSION VOLUME` command line is present, the element will be subdivided until a number of volumes each of roughly the specified volume is reached. A new particle will then be placed at the centroid of each of these sub-volumes.

If both the `MAX NUM PARTICLES` and `MIN PARTICLE CONVERSION VOLUME` commands are specified, the element will be subdivided until either the minimum volume of the sub-parts is reached, or the maximum number of sub-parts is reached. When one of these criteria is reached, a new particle is placed into each created sub-volume.

6.5.4.10 Active Periods

The following command lines can optionally be used in the `ELEMENT DEATH` command block:

```
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

These command lines determine when element death is active. See Section 2.6 for more information about these optional command lines.

6.5.5 Cohesive Zone Setup Commands

The commands listed here are all related to activation of cohesive zones by element death.

```
COHESIVE ZONE INITIALIZATION METHOD = <string>NONE|
ELEMENT STRESS AVG(NONE)
```



The `COHESIVE ZONE INITIALIZATION METHOD` command controls the initialization of cohesive zones that are activated through element death. This command should only be used if there are cohesive zones between elements and all nodes of those elements are initially attached together via multi-point constraints. When element death is used to deactivate the constraints (with the `DEATH METHOD = DEACTIVATE NODAL MPCS`), the exposed cohesive zone can be given an initial state. The options are to either do nothing (`NONE`) or to initialize the tractions in the cohesive element based on the stresses in the two elements on either side of the cohesive zone (`ELEMENT STRESS AVG`). How the cohesive zone uses the initial traction will depend on the cohesive surface material model used.

6.5.6 Example

The following example provides instructions to kill elements in `block_1` when they leave a bounding box. This type of element death can be useful in an analysis where some peripheral parts, because of fracture, separate and fly away from a central body, where this central body is the part of interest. In this case, these peripheral parts no longer impact the solution. The instructions in this `ELEMENT DEATH` command block will cause the parts to be killed, thus speeding up the computation.

```
begin element death out_of_bounds
  block = block_1
  # check x coordinates
  criterion is avg nodal value of coordinates(1) >= 10
  criterion is avg nodal value of coordinates(1) <= -10
  # check y coordinates
  criterion is avg nodal value of coordinates(2) >= 10
  criterion is avg nodal value of coordinates(2) <= -10
  # check z coordinates
  criterion is avg nodal value of coordinates(3) >= 10
  criterion is avg nodal value of coordinates(3) <= -10
end element death out_of_bounds
```

6.5.7 Element Death Visualization

When an element dies, information about this element will still be sent, along with information for all other elements, to the Exodus II results file. (Chapter 9 describes the output of element variables to the results file.) The death status of the elements may be output to the results file by requesting element variable output for the element variable `DEATH_STATUS`. Including the command line

```
ELEMENT DEATH_STATUS as death_var
```

in a `RESULTS OUTPUT` command block (Chapter 9) will output this element variable with the name `death_var` in the results file. The convention for `DEATH_STATUS` is as follows: An element with a value of 1.0 for `DEATH_STATUS` is a living element. An element with a value of less than or equal to 0.0 for `DEATH_STATUS` is a dead element.

In the case other than `MATERIAL CRITERION`, a `DEATH_STATUS` value between 1.0 and 0.0 indicates an element is in the process of dying. A dying element has its material stress scaled down

over a number of time steps. The current scaling factor for an element is given by `DEATH_STATUS`. Whether an element can have a value other than 0.0 or 1.0 for `DEATH_STATUS` will depend on whether the `DEATH STEPS` option has been used in the `ELEMENT DEATH` command block. If the number of steps over which death occurs is greater than 1, then `DEATH_STATUS` can be some value between 0.0 and 1.0.

If `DEATH_STATUS` is written to a results file, it is possible to use `DEATH_STATUS` to exclude killed elements from any view of the model in visualization software. A subset of the mesh showing only living elements can be created by visualizing only those elements for which `DEATH_STATUS > 0.0`. The procedure for filtering results in this way varies for different post-processing tools.

When an analysis is using element death, the log file contains a table of marker values that will be applied to dead elements. The marker values allow determining which elements were killed and by which criterion. The marker variables are stored in the element variable `KILLED_BY_CRITERION` and are available for output on the mesh results file. Additionally, a global count of how many elements were killed by each criterion is printed at the end of the run log file.



6.6 Particle Embedding

```
BEGIN PARTICLE EMBEDDING <string>particle_embedding_name
  BLOCK = <string>block_name
  INCLUDE ALL BLOCKS
  REMOVE BLOCK = <string>block_name
  PARTICLE SECTION = <string>section_name
  NEW MATERIAL = <string>material_name
    MODEL NAME = <string>model_name
  TRANSFER = <string>NONE|ALL (ALL)
  MAX NUM PARTICLES = <integer>num(1)
  MIN PARTICLE CONVERSION VOLUME = <real>vol
END [PARTICLE EMBEDDING]
```

Particle embedding allows the user to specify element blocks in which each element is embedded with particles during initialization, before the first time step is taken. This capability can remove the need to create particles during element death and may be used for particle-solid interactions.

The embedded particles are created in the same way as during particle conversion for element death, except that the original solid element remains alive after the particles are created. Particles that are created are constrained to move with a material point in their controlling element (i.e., the element in which they were created). This constraint is released only if the controlling element dies. In addition, these embedded particles may interact with non-embedded particles to produce internal forces on the embedded particles. These internal forces are transferred to the controlling element using the embedded particle's location in the element. Note that the mass of the particles embedded in an element is assigned as in particle conversion for element death but that the particles' mass does not affect the solution until the controlling element dies. Once a particle's controlling element dies it is freed and behaves as a particle element.

The commands for this block are discussed in Section [6.5.4.9](#) and work for particle embedding in the same way, save `TRANSFER`. Particle embedding has two opportunities for transferring element data: at initialization and at controlling element death. For particle embedding the `TRANSFER` command applies to both of these transfers. If the `ALL` option is used, an embedded particle will be initialized with the initial element data of its controlling element and will have its element data reset to the evolved element data of its controlling element when its controlling element dies. If the `NONE` option is used, the embedded particle's element data will be initialized as a non-embedded particle's would be, and the particle's element data will not be modified at controlling element death.

6.7 Explicitly Computing Derived Quantities

```
BEGIN DERIVED OUTPUT
  COMPUTE AND STORE VARIABLE =
    <string>derived_quantity_name
END DERIVED OUTPUT
```

The above command block is used to explicitly compute and store a derived quantity into an internal field. This is useful if the field is needed by an outside capability such as a transfer or error estimation.

For example, to use a derived quantity in a transfer, a `DERIVED OUTPUT` command block must be used. For instance, to transfer the von Mises stress norm:

```
BEGIN DERIVED OUTPUT
  COMPUTE AND STORE VARIABLE = von_mises
END DERIVED OUTPUT
```

Tables 9.9 through 9.12 in Section 9 list element variables available for different types of elements. Variables that are only computed at user request are designated with a `yes` in the `Derived` column of these tables. These are the quantities that must be listed in a `BEGIN DERIVED OUTPUT` command block if they are to be transferred during a coupled analysis.



6.8 Mesh Rebalancing

Mesh rebalancing is a feature in Presto that may improve the efficiency of an analysis. Two command blocks can be used to control mesh rebalancing: `REBALANCE` and `ZOLTAN PARAMETERS`. The `REBALANCE` command block is required; the `ZOLTAN PARAMETERS` command block is optional. Sections 6.8.1 and 6.8.2 describe these command blocks.

6.8.1 Rebalance

```
BEGIN REBALANCE
  ELEMENT GROUPING TYPE = SPLIT SPH AND STANDARD ELEMENTS |
    UNIFORM UNIFIED | SPLIT SPH AND DEM AND STANDARD ELEMENTS |
    SPLIT DEM AND STANDARD ELEMENTS |
    SPLIT PARTICLE AND STANDARD ELEMENTS
    (SPLIT SPH AND STANDARD ELEMENTS)
  INITIAL REBALANCE = ON|OFF(OFF)
  PERIODIC REBALANCE = ON|OFF|AUTO(OFF)
  REBALANCE STEP INTERVAL = <integer>step_interval
  LOAD RATIO THRESHOLD = <real>load_ratio
  COMMUNICATION RATIO THRESHOLD = <real>communication_ratio
  ZOLTAN PARAMETERS = <string>parameter_name
  DELETE DEACTIVATED ELEMENTS AFTER REBALANCE = ON|OFF(OFF)
  ACTIVE PERIODS = <string list>active_periods
  INACTIVE PERIODS = <string list>inactive_periods
END [REBALANCE]
```

Initial decomposition of a mesh for parallel runs with Sierra/SM is done by a utility called `decomp`. The initial decomposition provided by `decomp` may become sub-optimal as the simulation evolves, especially for cases involving large deformations. Therefore, Presto supports a simple mesh-rebalancing capability that can be used to improve the parallel performance of some problems. When mesh rebalancing is invoked, the parallel decomposition is changed, and elements are moved among the processors to balance the computational load and minimize the processor-to-processor communication. Mesh rebalancing may be useful in the following circumstances:

- The mesh decomposition produced by `decomp` for sphere elements (i.e., for SPH or peridynamics) can be poor. It is recommended that an initial mesh rebalance be done for all problems involving sphere elements.
- If a problem experiences very large deformations, periodic rebalancing may be helpful. In contact or SPH problems, communication is performed between physically nearby contact surfaces or SPH particles. To maintain optimum performance, it is helpful to have neighboring particles located on the same processors. Periodic mesh rebalancing can ensure that neighboring entities tend to remain on the same processor during large mesh deformations.

The `REBALANCE` command block is placed in the Presto region scope. The mesh rebalancing in Presto uses a mesh balancing library called Zoltan (Reference [15]). Zoltan performs the actual rebalancing. By default, Presto creates a Zoltan object with a default set of parameters. However, a Zoltan object with a customized set of parameters can be created and referenced from the `REBALANCE` command block.

6.8.1.1 Rebalance Command Lines

```
ELEMENT GROUPING TYPE = SPLIT SPH AND STANDARD ELEMENTS |  
    UNIFORM UNIFIED | SPLIT SPH AND DEM AND STANDARD ELEMENTS |  
    SPLIT DEM AND STANDARD ELEMENTS |  
    SPLIT PARTICLE AND STANDARD ELEMENTS  
    (SPLIT SPH AND STANDARD ELEMENTS)  
INITIAL REBALANCE = on|off (off)  
PERIODIC REBALANCE = on|off|auto (off)  
REBALANCE STEP INTERVAL = <integer>step_interval  
LOAD RATIO THRESHOLD = <real>load_ratio  
COMMUNICATION RATIO THRESHOLD = <real>ratio
```

The above command lines control how and when the rebalancing is done.

The `ELEMENT GROUPING TYPE` command line specifies how the elements are grouped for rebalancing. If set to `SPLIT SPH AND STANDARD ELEMENTS`, which is the default behavior, this command causes SPH and standard elements to be split into separate groups. If it is set to the `UNIFORM UNIFIED` option, all elements can be grouped together. This command also has options that relate to DEM an experimental element formulation. These DEM options are not recommended for general use.

The `INITIAL REBALANCE` command line is used to rebalance the mesh at time zero before any calculations occur. This option should be used if the initial mesh decomposition passed to Presto is poor.

If the `PERIODIC REBALANCE` COMMAND option is set to `on`, the mesh will be rebalanced every `step_interval` steps, where `step_interval` is the parameter specified by the `REBALANCE STEP INTERVAL` command line. If the option is `auto`, the mesh will be rebalanced every `step_interval` steps, when the communication ratio reaches a critical value, or when the load ratio reaches a critical value.

The communication ratio, currently defined only for SPH problems, is a measure of how much communication is required in the current mesh decomposition versus an estimate of the amount of communication with an optimal decomposition. Mesh rebalancing is expensive, so rebalancing should be done rarely. The `COMMUNICATION RATIO THRESHOLD` command line is used to specify the value of the communication ratio that triggers a rebalance. Setting this to a value between 1.25 and 1.5 is usually optimal.

The load ratio is the ratio of the current load (maximum number of elements on a processor divided by the average number of elements per processor). The `LOAD RATIO THRESHOLD` command specifies that a rebalance should occur when the load ratio exceeds the specified threshold. A specified value of 1.5 would trigger a rebalance when the load ratio is 50% higher than could be optimally obtained.

The `DELETE DEACTIVATED ELEMENTS AFTER REBALANCE` command line can be used to remove any inactive elements before performing the mesh rebalance. If a large number of elements have been deactivated due to element death, or due to use of time step sub-cycling, this may lead to a better load balance. With this option rebalance will get an optimal decomposition considering

only those elements that are currently alive and active and remove any other elements permanently from the data structure.

The `ACTIVE PERIODS AND INACTIVE PERIODS` commands can turn rebalancing on or off by time period. See Section 2.6 for more information regarding these commands.

6.8.1.2 Zoltan Command Line

The command line

```
ZOLTAN PARAMETERS = <string>parameter_name
```

references a `ZOLTAN PARAMETERS` command block named `parameter_name`. Various parameters for Zoltan can be set in the `ZOLTAN PARAMETERS` command block. If the `ZOLTAN PARAMETERS` command line is not provided, a default set of parameters is used. The default parameter command block is shown as follows:

```
BEGIN ZOLTAN PARAMETERS
  LOAD BALANCING METHOD = recursive coordinate bisection
    # string parameter
  OVER ALLOCATE MEMORY = 1.5 # real parameter
  REUSE CUTS = true # string parameter
  ALGORITHM DEBUG LEVEL = 0 # integer parameter
  CHECK GEOMETRY = true # string parameter
  ZOLTAN DEBUG LEVEL = 0 # integer parameter
END ZOLTAN PARAMETERS
```

See Section 6.8.2 for a discussion of the `ZOLTAN PARAMETERS` command block. Section 6.8.2 lists the command lines that can be used to set Zoltan parameters.

6.8.2 Zoltan Parameters

```
BEGIN ZOLTAN PARAMETERS
  LOAD BALANCING METHOD = <string>recursive coordinate bisection|
    recursive inertial bisection|Hilbert space filling curve
  DETERMINISTIC DECOMPOSITION = <string>false|true
  IMBALANCE TOLERANCE = <real>imb_tol
  OVER ALLOCATE MEMORY = <real>over_all_mem
  REUSE CUTS = <string>false|true
  ALGORITHM DEBUG LEVEL = <integer>alg_level
    # 0<=(alg_level)<=3
  CHECK GEOMETRY = <string>false|true
  KEEP CUTS = <string>false|true
  LOCK RCB DIRECTIONS = <string>false|true
  SET RCB DIRECTIONS = <string>do not order cuts|xyz|xzy|
    yzx|yxz|zxy|zyx
  RECTILINEAR RCB BLOCKS = <string>false|true
  RENUMBER PARTITIONS = <string>false|true
```

```

ZOLTAN DEBUG LEVEL = <integer>zoltan_level
# 0<=(zoltan_level)<=10
DEBUG PROCESSOR NUMBER = <integer>proc # 1<=proc
TIMER = <string> wall|cpu
DEBUG MEMORY = <integer>dbg_mem # 0<=(dbg_mem)<=3
END [ZOLTAN PARAMETERS]

```

The `ZOLTAN PARAMETERS` command block is used to set parameters for Zoltan (see Reference [15]), a program that can be used for mesh rebalancing in Presto. The `ZOLTAN PARAMETERS` command block is used in association with the `REBALANCE` command block. A `REBALANCE` command block may reference a `ZOLTAN PARAMETERS` command block via the name, `parameter_name`, for the parameter command block. Reference Section 6.8.1 regarding the use of the `ZOLTAN PARAMETERS` command block for mesh rebalancing in Presto.

Setting the parameters for Zoltan involves some understanding of how Zoltan works. Consult Reference [15] for a discussion of the parameters that can be set by the various command lines in the `ZOLTAN PARAMETERS` command block. Note that some of the command lines in this command block have comments that provide additional information about the parameters. The “#” symbol precedes a comment.

In the above command block, `=` and `IS` are the allowed delimiters, which is different from the usual Presto convention of `=`, `IS`, and `ARE`. Note that the `ZOLTAN PARAMETERS` command block should be specified in the `SIERRA` scope when it is referenced from the `ZOLTAN PARAMETERS` command line in the `REBALANCE` command block. When the default set of parameters is used for a Zoltan object, the `ZOLTAN PARAMETERS` command block need not be included in the input file.

6.9 Remeshing

```
BEGIN REMESH
  MAX REMESH STEP INTERVAL = <integer>stepInterval (Infinity)
  MAX REMESH TIME INTERVAL = <real>timeInterval (Infinity)
  NEW MESH MAX EDGE LENGTH RATIO = <real>newMaxRatio (1.25)
  NEW MESH MIN EDGE LENGTH RATIO = <real>newMinRatio (0.25)
  NEW MESH MIN SHAPE = <real>newShape (0.125)
  REMESH AT MAX EDGE LENGTH RATIO = <real>maxCutoffRatio
    (newMaxEdgeLengthRatio*1.75)
  REMESH AT MIN EDGE LENGTH RATIO = <real>minCutoffRatio
    (newMinEdgeLengthRatio*0.25)
  REMESH AT SHAPE = <real>cutoffShape (0.025)
  CONTACT CLEANUP = AUTO|OFF|ON (AUTO)
  DEBUG OUTPUT LEVEL = <integer>level (0)
  MAX REMESH REBALANCE METRIC = <integer>rebalanceMetric (1.25)
  MAX NUMBER ELEMENTS = <integer>maxNumElem (500000)
  MIN SHAPE IMPROVEMENT = <real>minShapeImprovement (0.001)
  MAX NUM PATCH RECREATIONS = <integer>maxNumPatchRecreations (5)
  POISON CRITICAL VALUE = <integer>poisonCriticalValue (4)
  POISON SUBTRACTION INTERVAL =
    <integer>poisonSubtractionInterval (10)
  MAX ITERATIONS = <integer>maxIterations (5)

  BEGIN REMESH BLOCK SET
    #
    # Parameters for remesh block set
    #
  END [REMESH BLOCK SET]

  BEGIN ADAPTIVE REFINEMENT
    #
    # Parameters for adaptive refinement
    #
  END [ADAPTIVE REFINEMENT]
END [REMESH]
```

The `REMESH` command block, which is used within the region scope, sets parameters for remeshing a portion of the mesh. Remeshing involves removing badly shaped elements and inserting new elements of better quality that occupy the same volume. Depending on the degree to which the original elements are deformed, the new elements may occupy slightly more or slightly less volume than the original mesh. If regions of the mesh cannot be meshed with well-shaped elements having reasonable time steps, they may be removed entirely, potentially changing the topology. Examples of such regions include exterior slivers or thin parts.

Remeshing is used exclusively with node-based tetrahedra. Whenever possible, remeshing avoids moving, deleting or adding nodes. The node-based tetrahedron stores both material and kinematic state information at the nodes and thus avoids many of the state advection problems associated with

remeshing. However, some amount of state advection or non-conservation of energy, momentum, and mass may occur during remeshing.

In addition to remeshing locations with badly shaped elements, remeshing can be used to adaptively refine a mesh. Adaptive refinement may be applied to an initially uniform mesh to produce one with a large number of elements in a region of interest and a coarse mesh elsewhere. Mesh refinement is fully conforming and works by altering the nodal characteristic edge lengths in an adapted mesh. The remeshing algorithm attempts to make the new mesh density consistent with the adapted characteristic edge lengths. Mesh adaptation happens only during remesh steps. Specifying a remesh step or a remesh time interval may be used to trigger adaptive refinement even if no large element deformation is taking place.

6.9.1 Remeshing Commands

The following commands control how and when remeshing is performed:

```

MAX REMESH STEP INTERVAL = <integer>stepInterval (Infinity)
MAX REMESH TIME INTERVAL = <real>timeInterval (Infinity)
NEW MESH MAX EDGE LENGTH RATIO = <real>newMaxRatio (1.25)
NEW MESH MIN EDGE LENGTH RATIO = <real>newMinRatio (0.25)
NEW MESH MIN SHAPE = <real>newShape (0.125)
REMESH AT MAX EDGE LENGTH RATIO = <real>maxCutoffRatio
    (newMeshMaxEdgeLengthRatio*1.75)
REMESH AT MIN EDGE LENGTH RATIO = <real>minCutoffRatio
    (newMeshMinEdgeLengthRatio*0.25)
REMESH AT SHAPE = <real>cutoffShape (0.025)
CONTACT CLEANUP = AUTO|OFF|ON (AUTO)
DEBUG OUTPUT LEVEL = <integer>level (0)
MAX REMESH REBALANCE METRIC = <integer>rebalanceMetric (1.25)
MAX NUMBER ELEMENTS = <integer>maxNumElems (500000)
MIN SHAPE IMPROVEMENT = <real>minShapeImprovement (0.001)
MAX NUM PATCH RECREATIONS = <integer>maxNumPatchRecreations (5)
POISON CRITICAL VALUE = <integer>poisonCriticalValue (4)
POISON SUBTRACTION INTERVAL =
    <integer>poisonSubtractionInterval (10)
MAX ITERATIONS = <integer>maxIterations (5)

```

The `MAX REMESH STEP INTERVAL` command defines the maximum number of simulation time steps that can pass between remeshing steps. Note that if any value of `MAX REMESH STEP INTERVAL` is specified, the code will perform a remesh step during initialization. This is useful for doing an initial mesh adaptation.

The `MAX REMESH TIME INTERVAL` command defines the maximum simulation time that can pass between remeshing steps.

The `NEW MESH MAX EDGE LENGTH RATIO`, `NEW MESH MIN EDGE LENGTH RATIO`, and `NEW MESH MIN SHAPE` commands specify target values for maximum edge length ratio, minimum edge length ratio, and minimum mesh shape in the new mesh. These metrics are described below in the definitions of the `REMESH AT MAX EDGE LENGTH RATIO`, `REMESH AT MIN EDGE`

LENGTH RATIO, and REMESH AT SHAPE commands, respectively. It is recommended that the thresholds for triggering remeshing be at least a factor of four smaller than the new mesh values to prevent excessive remeshing calls.

The REMESH AT MAX EDGE LENGTH RATIO command specifies a threshold edge length ratio at which remeshing is triggered. The ratio is determined as the ratio of the longest edge connected to a given node to the characteristic initial mesh edge length associated with that node. Edge lengths that become large may no longer be able to represent required geometric details or accurately compute contact. The remeshing algorithm attempts to maintain edge lengths at roughly their original lengths, which may vary significantly across the mesh.

The REMESH AT MIN EDGE LENGTH RATIO command is similar to the REMESH AT MAX EDGE LENGTH RATIO command. This command triggers remeshing when the ratio of the shortest edge connected to a given node to the characteristic initial mesh edge length associated with that node falls below the specified threshold value. If edge lengths become too small, the mesh may have an excessively small time step or too many elements.

The REMESH AT SHAPE command defines the critical shape at which refinement is triggered. The element shape metric is based on a minimum tetrahedron face angle. The optimum tetrahedron has four equilateral faces and angles between those faces of approximately 50 degrees. A degenerate or inverted tetrahedron has a minimum face angle less than or equal to zero degrees. The element shape provides a measure of how close an element is to becoming degenerate or inverted.

The CONTACT CLEANUP command determines whether contact-related algorithms are run following remeshing. Remeshing near areas of contact may result in violations of contact constraints due to changes in element topology. Reapplying contact algorithms following remeshing ensures that contact requirements are maintained.

The DEBUG OUTPUT LEVEL command controls the amount of remeshing-related output printed to the log file and terminal output. If this is set to zero, no output occurs, while larger numbers result in larger amounts of output. Output may include:

- Step-by-step values of metrics that trigger remeshing.
- The number of elements changed in a remesh step.
- Mass, energy, and momentum conservation balances during remeshing.

The MAX REMESH REBALANCE METRIC command line is used to control when the load rebalancing algorithm is called. During remeshing, the number of elements per processor can become out of balance, causing the parallel analyses to slow down. The specified `rebalance_metric` is a threshold for the ratio of the maximum number of elements on any processor to the average number of elements per processor. After each remeshing is performed, if that ratio exceeds the specified `rebalance_metric`, the rebalancing algorithm is called. The default value for this parameter is 1.25.

The MAX NUMBER ELEMENTS command line is used to set the maximum number of elements per processor that may be created during a remeshing analysis. If this value is exceeded, a warning is issued and no further remeshing is done. The default value for this parameter is 500000.

The MAX ITERATIONS command line is used to set the maximum number of remesh iterations taken in a single time step. Lower values cause remesh to possibly run faster at the expense of

worse elements or deleting difficult regions of the mesh. Valid values are 1 to infinity. The default value is 5.

The `MAX NUM PATCH RECREATIONS` command line is used to set the maximum number of patch recreations. This is an advanced remeshing algorithm option that relates to the iteration loop around remesh/parallel consolidation/overlap removal. Lower values cause remeshing to possibly run faster at the expense of quality elements (similar to `MAX ITERATIONS` described above). Valid values are 1 to infinity. The default value is 5.

The `MIN SHAPE IMPROVEMENT` command line is used to set what is considered a valid remesh operation. Generally larger values will cause remesh to function faster at expense of less overall mesh quality or more element deletion. Valid values are from 0.0 to approximately 0.1

The commands `POISON CRITICAL VALUE` and `POISON SUBTRACTION INTERVAL` are used to mark bad mesh regions. Each time an element is marked as bad all its nodes get a poison counter. Every subtraction interval steps all nodes remove one poison counter (never going below zero). If a node ever has more than the critical value of poison counters the node and all elements attached to it are removed from the mesh. This control is used to remove parts of the mesh that are frequently remeshing, likely due to a poorly behaving material model. Decreasing the critical value or increasing the subtraction interval cause the algorithm to more aggressively delete parts of the mesh. Increasing the critical value or decreasing the subtraction interval cause the code to avoid element deletion. Valid critical value 2 to Infinity. Valid subtraction interval 1 to Infinity. Note, setting subtraction interval to 1 in effect turns off the poison node element deletion.

6.9.2 Remesh Block Set

```
BEGIN REMESH BLOCK SET
  BLOCK = <string list>block_names
  REMOVE BLOCK = <string list>block_names
  INCLUDE ALL BLOCKS
END [REMESH BLOCK SET]
```

The `BEGIN REMESH BLOCK SET` command block, used in the scope of the `REMESH` command block, defines the element block or set of blocks that the given remesh command block applies to. The elements in the element block will be checked against the shape criteria and may be affected by remeshing. Note that it is currently assumed that all elements to be remeshed are part of a single material. Remeshing of equivalenced mesh blocks is not well supported.

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of blocks. See Section 7.1.1 for more information about the use of these command lines for creating a set of blocks used by the command block. There must be at least one `BLOCK` or `INCLUDE ALL BLOCKS` command line in the command block.

The `REMOVE BLOCK` command line allows the user to delete blocks from the set specified in the `BLOCK` and/or `INCLUDE ALL BLOCKS` command line(s) through the string list `block_names`. The `REMOVE BLOCK` command line is typically used with the `INCLUDE ALL BLOCKS` command line. To include all but a few of the element blocks, a combination of the `REMOVE BLOCK` command line and `INCLUDE ALL BLOCKS` should minimize input information.

6.9.3 Adaptive Refinement

```
BEGIN ADAPTIVE REFINEMENT
#
# adaptive refinement control commands
ADAPT TYPE = NODE_PROXIMITY|POINT_PROXIMITY|
    SHARP_EDGE_PROXIMITY|FACE_PROXIMITY
RADIUS = <real>radius
ADAPT SHARP ANGLE = <real>angle (45)
ADAPT SMOOTH ANGLE = <real>angle (10)
GEOMETRIC POINT COORDINATES = <real>x <real>y <real>z
ADAPT SIZE RATIO = <real>ratio
TARGET MESH SIZE = <real>target_size
#
# tool mesh entity commands
NODE SET = <string list>nodelist_names
REMOVE NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
REMOVE SURFACE = <string list>surface_names
BLOCK = <string list>block_names
REMOVE BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
#
# activation commands
ACTIVE PERIODS = <string>period_names
INACTIVE PERIODS = <string>period_names
END [ADAPTIVE REFINEMENT]
```

The `ADAPTIVE REFINEMENT` command block, used in the context of the `REMESH` command block, contains commands related to adaptive refinement. Adaptive refinement can be useful when the desired level of refinement in a region of the mesh is unknown when the mesh is created, or when a refinement region should move during the analysis. For example, complex physics may occur near moving contact boundaries. Adaptive refinement can be used to produce a fine mesh only near these boundaries and then have that refined region move with the contact boundaries as they move.

6.9.3.1 Adaptive Refinement Control Commands

The following commands used within the `ADAPTIVE REFINEMENT` command block control the adaptive refinement algorithm:

```
ADAPT TYPE = NODE_PROXIMITY|POINT_PROXIMITY|
    SHARP_EDGE_PROXIMITY|FACE_PROXIMITY
RADIUS = <real>radius
ADAPT SHARP ANGLE = <real>angle (45)
ADAPT SMOOTH ANGLE = <real>angle (10)
GEOMETRIC POINT COORDINATES = <real>x <real>y <real>z
```



```
ADAPT SIZE RATIO = <real>ratio  
TARGET MESH SIZE = <real>target_size
```

The `ADAPT TYPE` command specifies the type of refinement. Currently four options exist:

- The `NODE_PROXIMITY` option refines areas of the remesh blocks that are near nodes of the tool blocks.
- The `SHARP_EDGE_PROXIMITY` option refines areas of the remesh blocks that are near exterior sharp edges of tool blocks. This command is specialized to refine regions for contact. To correctly represent contacting geometries, small elements are often required to allow material flow around sharp intruding edges.
- The `POINT_PROXIMITY` option refines all nodes near a defined point in space.
- The `FACE_PROXIMITY` option refines regions of the remesh blocks that are near faces of the tool blocks.

The `RADIUS` command defines the minimum distance from a node to a given tool object in the remesh block for that node to be considered for refinement. For example, if the refinement type is `POINT_PROXIMITY`, each node within a distance of `RADIUS` to the defined point will be refined. If refinement type is `NODE_PROXIMITY`, each node within a distance of `RADIUS` to any node in the tool block will be refined. Nodes that are closer to the refinement point have more refinement applied to them than nodes that are farther away. Thus, refinement grades the mesh smoothly from fine to coarse elements over the distance `RADIUS`.

The `ADAPT SHARP ANGLE` and `ADAPT SMOOTH ANGLE` commands are used only with the `SHARP_EDGE_PROXIMITY` refinement type and define edge sharpness thresholds for full, partial, and no mesh refinement. Edges with angles sharper (smaller) than `SHARP ANGLE` trigger full mesh refinement. Edges with angles smoother (larger) than `ADAPT SMOOTH ANGLE` do not trigger mesh refinement. Edges with angles that fall between these two angles trigger a reduced level of mesh refinement. These two commands define a transition zone over which the level of mesh refinement varies smoothly from full refinement to zero refinement.

The `GEOMETRIC POINT COORDINATES` command is used only with the `POINT_PROXIMITY` refinement type. This command defines the spatial location of the refinement tool point. This command may be used multiple times to define multiple refinement points.

The `ADAPT SIZE RATIO` command defines the degree of refinement applied. A value of 2.0 would specify that the characteristic element size should be reduced by a factor of two in the refinement zone. A value less than one specifies that the mesh should be coarsened. Alternatively, the `TARGET MESH SIZE` may be used to directly specify the mesh size in the zone of adaptive refinement rather than specifying that size relative to the original mesh size.

6.9.3.2 Tool Mesh Entity Commands

The `NODE_PROXIMITY` and `SHARP_EDGE_PROXIMITY` types of adaptive refinement are based on proximity to tool mesh objects, which are collections of nodes or edges. For example, in a simulation of a sheet metal forming process, the punch tool may be defined as a tool mesh object, thus triggering adaptive refinement in the sheet metal that is in proximity to the punch tool.

These tool objects do not refine the body to be refined, but define objects that will trigger refinement

if they are in the proximity of the body to be refined. The following commands are used within the `ADAPTIVE REFINEMENT` command block to define tool entities:

```
NODE SET = <string list>nodelist_names
REMOVE NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
REMOVE SURFACE = <string list>surface_names
BLOCK = <string list>block_names
REMOVE BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
```

These commands are a set of Boolean operators for constructing a set of mesh entities. See Section 7.1.1 for more information about the use of these command lines. Multiple types of these commands can be used together to form a set of nodes or edges.

Any type of mesh entity can be used for the `NODE_PROXIMITY` adaptivity type. A set of nodes is formed from those contained in all node sets, surfaces, and blocks listed. For the `SHARP_EDGE_PROXIMITY` adaptivity type, however, only blocks and surfaces may be used. That adaptivity type requires information about element edges that is not available from node sets.

6.9.3.3 Activation Commands

Adaptive refinement, if used, is by default active throughout an analysis. If desired, it can optionally be activated or deactivated during select time periods using the following commands in the `ADAPTIVE REFINEMENT` command block:

```
ACTIVE PERIODS = <string>period_names
INACTIVE PERIODS = <string>period_names
```

See Section 2.6 for more information regarding these commands.

6.9.4 Known Issues

Though remeshing is considered a production capability, the automated mesh generation process is sometimes not fully robust. Small holes that fail to mesh may show up in the remeshed body over time. Sometimes these holes can cause substantial stability problems.

The remeshing algorithm is generally incompatible with equation of state (EOS) material models as the mechanisms to map volumetric material terms after remeshing have not been implemented at this time.

References

- [1] L.M. Taylor and D.P. Flanagan. Pronto3D: A three-dimensional transient solid dynamics program. Technical Report SAND87-1912, Sandia National Laboratories, Albuquerque, NM, March 1989. [pdf](#).

- [2] M.M. Rashid. Incremental kinematics for finite element applications. *International Journal for Numerical Methods in Engineering*, 36:3937–3956, 1993. [doi](#).
- [3] C.R. Dohrman, M.W. Heinstein, J. Jung, S.W. Key, and W.R. Witkowski. Node-based uniform strain elements for three-node triangular and four-node tetrahedral meshes. *International Journal for Numerical Methods in Engineering*, 47:1549–1568, 2000. [doi](#).
- [4] S.W. Key and C.C. Hoff. An improved constant membrane and bending stress shell element for explicit transient dynamics. *Computer Methods in Applied Mechanics and Engineering*, 124(1–2):33–47, 1995. [doi](#).
- [5] T.A. Laursen, S.W. Attaway, and R.I. Zadoks. SEACAS theory manuals: Part III. finite element analysis in nonlinear solid mechanics. Technical Report SAND98-1760/3, Sandia National Laboratories, Albuquerque, NM, 1999. [pdf](#).
- [6] D.P. Flanagan and T. Belytschko. A uniform strain hexahedron and quadrilateral with orthogonal hourglass control. *International Journal for Numerical Methods in Engineering*, 17:679–706, 1981. [doi](#).
- [7] J.W. Swegle. SIERRA: PRESTO theory documentation: Energy dependent materials version 1.0. Technical Report SAND2009-3801P, Sandia National Laboratories, Albuquerque, NM, October 2001.
- [8] Q. Yang, A. Mota, and M. Ortiz. A class of variational strain-localization finite elements. *International Journal for Numerical Methods in Engineering*, 62:1013–1037, 2005. [doi](#).
- [9] P. Madabhushi-Raman and J.F. Davalos. Static shear correction factor for laminated rectangular beams. *Composites Part B: Engineering*, 27(3):285–293, 1996. [doi](#).
- [10] G.D. Sjaardema. Overview of the Sandia National Laboratories engineering analysis code access system. Technical Report SAND92-2292, Sandia National Laboratories, Albuquerque, NM, January 1993. [pdf](#).
- [11] J.W. Swegle, S.W. Attaway, M.W. Heinstein, F.J. Mello, and D.L. Hicks. An analysis of smoothed particle hydrodynamics. Technical Report SAND93-2513, Sandia National Laboratories, Albuquerque, NM, March 1994. [pdf](#).
- [12] J. Monaghan. SPH without a tensile instability. *Journal of Computational Physics*, 159(2): 290–311, 2000. [doi](#).
- [13] JT Ostien, JW Foulk, A Mota, and MG Veilleux. A 10-node composite tetrahedral finite element for solid mechanics. *International Journal for Numerical Methods in Engineering*, 2016.
- [14] K.H. Brown, R.M. Summers, M.W. Glass, A.S. Gullerud, M.W. Heinstein, and R.E. Jones. ACME: Algorithms for contact in a multiphysics environment, API version, 2.2. Technical Report SAND2004-5486, Sandia National Laboratories, Albuquerque, NM, October 2001. [pdf](#).

- [15] K. Devine, E. Boman, R. Heapby, B. Hendrickson, and C. Vaughan. Zoltan data management service for parallel dynamic applications. *Computing in Science and Engineering*, 4:90–97, 2002. [doi](#). See also the ZOLTAN web site. [link](#).

Chapter 7

Boundary Conditions and Initial Conditions

Sierra/SM offers a variety of options for defining boundary and initial conditions. Typically, boundary and initial conditions are defined on some subset of mesh entities (node, element face, element) defining a model. Sierra/SM offers a flexible means to define subsets of mesh entities. Section 7.1.1 describes commands that will let the user define some subset of a mesh entity using a collection of commands that constitute a set of Boolean operators.

The remaining parts of this chapter discuss the following functionality:

- Section 7.3 presents methods for setting the initial values of variables in Sierra/SM. Sierra/SM has the flexibility to set a complex initial state for some variable such as nodal velocity or element stress.
- Kinematic boundary conditions typical of a solid mechanics code (fixed displacement, prescribed acceleration, etc.) are options in Sierra/SM and described in Section 7.4. Most of these boundary conditions allow the specification of a time history using a function, a user subroutine, or by reading values from a mesh file.
- Section 7.5 documents many initial velocity options available in Sierra/SM.
- Force boundary conditions typical of a solid mechanics code (prescribed force, traction, etc.) are options in Sierra/SM and described in Section 7.6. Most of these force boundary conditions allow the specification of a time history using a function or a user subroutine.
- Section 7.8 discusses the gravity load option. A gravity load is a body force boundary condition.
- Section 7.9 provides a description of centripetal force boundary conditions can be applied about a prescribed axis.
- Section 7.10 details many options available for describing a temperature field in Sierra/SM.
- Section 7.11 details the options available for describing a pore pressure field.
- Section 7.12 documents how fluid pressure boundary conditions can be applied as a hydrostatic pressure to each node pertaining to a set of prescribed surfaces.
- Section 7.13 documents how the inertia relief boundary condition can be applied to counterbalance the other force boundary conditions on the specified blocks, surfaces, and/or node-sets.
- Section 7.14 describes many specialized boundary conditions.

7.1 General Boundary Condition Concepts

There are general principles that apply to all the available types of boundary conditions. To apply a boundary condition, a set of mesh entities and the magnitude and/or direction in which it is to be applied must be specified. Sierra/SM provides several methods for both specifying the set of mesh entities and for prescribing how the boundary condition is to be applied. The general concepts on how this is done are applicable to all the boundary condition types, and are described in the following sections.

7.1.1 Mesh-Entity Assignment Commands

A number of standard command lines exist to define a set of mesh entities (node, element face, element) associated with some type of boundary, initial, or load condition. All these command lines exist within the command blocks for the various prescribed conditions, which in turn exist within the region scope. These command lines, taken collectively, constitute a set of Boolean operators for constructing sets of mesh entities.

The first set of command lines we will consider is as follows:

```
NODE SET = <string list>nodelist_names
NODE      = <integer list>node_ids

SURFACE = <string list>surface_names

BLOCK = <string list>block_names
ELEMENT = <integer list>element_ids

RIGID BODY = <string list>rigid_body_names

INCLUDE ALL BLOCKS
```

In the above command lines, the string list `nodelist_names` is used to represent one or more node sets as discussed in Section 1.6. A node set is referenced as `nodelist_id`, where `id` is some integer. For example, suppose there are three node lists in a model: 10, 23, and 105. To combine all these node lists so that they form one set of nodes for, say, a boundary condition or initial condition, the user would invoke the command line

```
NODE SET = nodelist_10 nodelist_23 nodelist_105
```

This convention applies as well to any surface-related command line that uses the string list `surface_names`, any block-related command line that uses the string list `block_names`, and any rigid body command line that uses the string list `rigid_body_names`.

The `NODE SET` command line associates a set of nodes with an initial, boundary, or load condition. A condition may be applied to multiple node sets by putting multiple node set names on the command line or by repeating the command line multiple times. The `NODE SET` line may not be

used to apply kinematic boundary conditions to a rigid body; a `BLOCK` or `RIGID BODY` line must be used.

The `NODE` line adds individual nodes to the boundary by condition by their nodal ID in the input mesh. Multiple nodes may be specified on one line or this command may be specified multiple times. The `NODE` line may not be used to apply kinematic boundary conditions to a rigid body; a `BLOCK` or `RIGID BODY` line must be used.

The `ELEMENT` line adds individual elements to the boundary by condition by their element ID in the input mesh. Multiple elements may be specified on one line or this command may be specified multiple times. The `ELEMENT` line may not be used to apply kinematic boundary conditions to a rigid body; a `BLOCK` or `RIGID BODY` line must be used.

The `SURFACE` command line associates a set of element faces or their attached nodes with a boundary, initial, or load condition. A condition may be applied to multiple surfaces by putting multiple surface names on the command line or by repeating the command line multiple times. For example, suppose we wish to use the fixed displacement kinematic boundary condition. Although this is a nodal boundary condition (the condition is applied to individual nodes), a `SURFACE` command line can be used to establish the set of nodes. If the command line

```
SURFACE = surface_101
```

appears in a fixed displacement boundary condition, then all the nodes associated with surface 101 will be associated with the boundary condition. The `SURFACE` line may not be used to apply kinematic boundary conditions to a rigid body; a `BLOCK` or `RIGID BODY` line must be used.

The `BLOCK` command line associates a set of elements and its nodes and faces with a boundary condition. A boundary condition may be applied to multiple blocks by putting multiple block names on the command line or by repeating the command line multiple times.

For example, suppose we wish to use the fixed displacement kinematic boundary condition as in the previous example. If the command line

```
BLOCK = block_50
```

appears in a fixed displacement kinematic boundary condition, then all the nodes associated with block 50 will be associated with the boundary condition.

The `RIGID BODY` command line associates the boundary condition with a rigid body. The rigid body must have been defined in the input file within a `BEGIN RIGID BODY` command block. See Section 6.3.1 for an explanation of how to create rigid bodies.

Either a `BLOCK` or a `RIGID BODY` command line must be used for kinematic boundary conditions on rigid bodies. If the `block_name` specified in the `BLOCK` line has been defined as a rigid body, or if the `RIGID BODY` is used, the specified kinematic condition will be applied to the rigid body reference node.

The `INCLUDE ALL BLOCKS` command line associates all blocks and hence all elements and nodes in the model with a boundary, initial, or load condition. The block command lines associated with boundary conditions, initial conditions, and gravity will NOT generate surfaces.

Any combination of the above command lines can be used to create a union of mesh entities. Suppose, for example, that the command lines

```
NODE SET = nodelist_2
SURFACE = surface_3
NODE = 7 8 15
ELEMENT = 31
```

appear in a `FIXED DISPLACEMENT` command block for a kinematic boundary condition. The set of nodes associated with the boundary condition will be the union of the set of nodes associated with surface 3, the set of nodes associated with node set 2, the nodes with ids 7, 8 and 15, and the nodes of element 31.

When a union of mesh entities is created by using two or more of the above command lines, a node or element face may appear in more than one node set, surface or block. However, the prescribed condition is applied to each node or face only once. For example, node 67 may be a part of nodelist 2 and surface 3 but the boundary condition will only be applied to node 67 once.

The set of mesh entities associated with a boundary, initial, or load condition can be edited (mesh entities can be deleted from the set) by using the following command lines:

```
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

The `REMOVE NODE SET` command line deletes the nodes in the specified node set from the set of nodes used by the condition.

The `REMOVE SURFACE` command line deletes a set of element faces and their associated nodes from the set of element faces used by the prescribed condition.

The `REMOVE BLOCK` command line deletes a set of elements and their associated nodes from the set of elements used by the prescribed condition.

7.1.2 Methods for Specifying Boundary Conditions

There are three main methods which can be used to prescribe most types of boundary conditions available in Sierra/SM.

- The boundary condition can be prescribed using commands in the input file. These commands are categorized as “specification commands” in this document. Depending on the type of the boundary condition, it is necessary to prescribe its direction and/or magnitude. Boundary conditions can be specified this way when a set of mesh entities is to experience a similar condition with a time variation that can be expressed by a function. One of the following commands is used to specify the direction of the boundary condition: `COMPONENT`, `DIRECTION`, `CYLINDRICAL AXIS`, or `RADIAL AXIS`. The magnitude is defined using one of `MAGNITUDE`, `FUNCTION` or `ANGULAR VELOCITY`. These commands are used in various combinations depending on the type of the boundary condition. The details of how to use them are provided in the descriptions of the various boundary condition types.

- If the nature of the boundary condition is such its variation in time and space cannot be described easily by the combination of a function and a direction, it may be necessary to use a user-defined subroutine. User subroutines provide a general capability to define how kinematic or force boundary conditions are applied. The use of user-defined subroutines does increase the complexity of defining the model, however. The user must write and debug the subroutine and compile and link it in with Sierra/SM. Because of the added complexity, user subroutines should only be used if the needed capability is not provided by the other methods of prescribing boundary conditions.
- For some types of boundary conditions, the values of the field to be prescribed can be read in from an existing output database. This is often used as a method to transfer results from one analysis code to another. One of the common uses for this capability is to compute temperatures using a thermal code, and then transfer the temperature fields to Sierra/SM to study combined mechanical and thermal effects. This capability can be used either to read in initial values or to read in a series of values that vary over time.

In the following sections describing specific types of boundary conditions, the commands are grouped according to these three categories.

7.1.3 External Mesh Database Commands

```

READ VARIABLE = <string>read_var_name
TIME = <real>time|FIRST|LAST
COPY VARIABLE = <string>copy_var_name
    [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)

```



Warning: At this time the `COPY VARIABLE` command can only be used with nodal variables. Neither the `COPY VARIABLE` command nor the `READ VARIABLE` command is compatible with global variables.

A number of boundary conditions work by applying some field defined on either the input mesh database or an external mesh database. For example a prescribed temperature may be applied by reading the spatial time history of temperature from a mesh file. The mesh files are defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1.

The `READ VARIABLE` command is used to read data from the input mesh for the region. The `read_var_name` string specifies the name of the variable as it appears on the mesh database. Generally the size and types of variables that are read need to be compatible with the types of target fields they are defining. Component variable syntax may also be used to help resolve ambiguity, see Section 9.1.

The field to be read may be defined at a number of different time slices on the mesh file. The default behavior is interpolate these values to the current simulation time to extract the value of the field to transfer. The `TIME` command line can optionally be used to select a specific time from which to read the variable from. If necessary, values are interpolated to the specified time. In addition to specifying a time, the keyword `FIRST` or `LAST` may be used to read values corresponding to the

first or last time step in the input mesh.

The `READ VARIABLE` command may be used in conjunction with `SCALE FACTOR` or `FUNCTION` commands, in such cases these source terms are multiplied together.

The copy variable function more flexible version of the read variable command. However, the copy variable command only works for nodal variables. The `COPY VARIABLE` command may be used for the following can be used for the following purposes:

- if the source field is defined on the input mesh file, but the analysis will be doing some extensive topology modifications, the copy variable command can be used to map the field that exists on the original geometry to the new geometry. This is necessary when performing such operations as dynamic mesh re-balancing, particle creation, or XFEM geometry modification.
- if the field is defined on some mesh database other than the input mesh file this command may be used to automatically map data between the two meshes databases. For example the temperature could have been calculated via an Aria run on one mesh, and then applied to the same geometry, but different mesh topology in Sierra/SM. For this mapping to function properly either the two meshes must occupy the same volume (`MAP_BY_PROXIMITY`) or the meshes must have identical topology and node and element ID numbers (`MAP_BY_ID`).

This `COPY VARIABLE` command specifies that the variable named `copy_var_name` will be used as the source variable. This `copy_var_name` is the name of a nodal field on an input mesh database. The `FROM MODEL <string>model_name` portion of the command is optional. If it is specified, the results are read from the mesh database named `model_name`, see Section 6.1. If the `model_name` is not provided, the region's finite element mesh database will be used to read the source variable.

Two types of mapping are available. The default `MAP_BY_PROXIMITY` option can be used to map data from a model with that occupies the same volume, but has a different topology than the analysis model. When using `MAP_BY_PROXIMITY` a node in the target mesh will define data by interpolating from the node in the source mesh at the closest location. This mapping is done in the original coordinates and is not affected by displacement. The option `MAP_BY_ID` can be used to directly send data between nodes in the source and target mesh with the exact same ID numbers. The `MAP_BY_ID` option can provide a faster and potentially more accurate mapping if the two models being mapped between are known to have completely identical topology.

Examples

The following command would initialize the `unrotated_stress` in the model based on the “stress” values that existed in the input mesh file at time 1.5.

```
BEGIN INITIAL CONDITION
  INITIALIZE VARIABLE NAME = unrotated_stress
  VARIABLE TYPE = ELEMENT
  READ VARIABLE = stress
  TIME = 1.5
END
```

The following command would define the temperature in a block as a function of time based on the state variable at index five of a state array defined in the input mesh.

```
BEGIN PRESCRIBED TEMPERATURE
  block = block_32
  READ VARIABLE = aria_state_output(5)
END
```

The following command would map the displacement defined in some other mesh to the boundary of the analysis mesh. Note the “dispOut” variable must be a three component vector field on the mesh on which it is defined.

```
BEGIN PRESCRIBED DISPLACEMENT
  node set = nodelist_1 nodelist_50
  COPY VARIABLE = dispOut FROM MODEL dispSourceModel \#
  MAP_BY_PROXIMITY
END
```

7.2 Initial Mesh Modification

```
BEGIN INITIAL MESH MODIFICATION
BLOCK = <string list>block_name
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_name
ROTATE ANGLE <real>angle
    ABOUT ORIGIN <real>Ox <real>Oy <real>Oz
    DIRECTION <real>Dx <real>Dy <real>Dz
    MOVE X <real>Mx Y <real>My Y <real>Mz

    ROTATE BLOCK <string list>block_names
        ANGLE <real>angle
        ABOUT ORIGIN <real>Ox <real>Oy <real>Oz
        DIRECTION <real>Dx <real>Dy <real>Dz
    MOVE BLOCK <string list>block_names
        X <real>Mx Y <real>My Y <real>Mz
END
```

A handful of transformations can be applied to the initial mesh from within the input deck. These transformations happen before any other operations being performed such as contact overlap removal or evaluating initial conditions. Initial mesh modification will only affect the initial mesh read into the file and ignore anything created during the run.

The `ROTATE` command will rotate the blocks set by `INCLUDE ALL BLOCKS` or `BLOCK`. The rotation of these blocks is defined by a rotation axis and angle of rotation about that axis. The origin and direction of the rotation axis are defined by the `ABOUT ORIGIN` and `DIRECTION` commands, respectively. The `ANGLE` command sets the rotation angle, in degrees, by following the right-hand rule.

The `MOVE` command will translate the blocks set by `INCLUDE ALL BLOCKS` or `BLOCK` with the specified `X`, `Y`, and `Z` translation vector.

The `ROTATE BLOCK` command will rotate the named set of blocks, `block_names`. The rotation of these blocks is defined by a rotation axis and angle of rotation about that axis. The origin and direction of the rotation axis are defined by the `ABOUT ORIGIN` and `DIRECTION` commands, respectively. The `ANGLE` command sets the rotation angle, in degrees, by following the right-hand rule.

The `MOVE BLOCK` command will translate the named set of blocks with the specified `X`, `Y`, and `Z` translation vector.

The syntax and behavior of the mesh modification commands mimic those in Cubit. The defined transformations will be evaluated in order. Thus moving a block then rotating it about an axis will give a different result from rotating the block first then moving it.

7.3 Initial Variable Assignment

```
BEGIN INITIAL CONDITION
#
# mesh-entity set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# variable identification commands
INITIALIZE VARIABLE NAME = <string>var_name
VARIABLE TYPE = NODE|EDGE|FACE|ELEMENT|GLOBAL
#
# specification command
MAGNITUDE = <real list>initial_values
#
# Weibull probability distribution commands
WEIBULL SHAPE = <real>shapeVal
WEIBULL SCALE = <real>scaleVal
WEIBULL MEDIAN = <real>medianVal
WEIBULL SEED = <int>seedVal(123456)
WEIBULL SCALING FIELD TYPE = NODE|EDGE|FACE|ELEMENT|
    GLOBAL(ELEMENT)
WEIBULL SCALING FIELD NAME = <string>fieldVal(VOLUME)
WEIBULL SCALING REFERENCE VALUE = <real>value
WEIBULL SCALING EXPONENT SCALE = <real>scaleVal(1.0)
#
# external database commands
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID(MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name |
    SURFACE SUBROUTINE = <string>subroutine_name |
    ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF|ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

```

#
# distance function commands
DISTANCE TYPE = XTYPE|YTYPE|ZTYPE|RTYPE|
    RXTYPE|RYTYPE|RZTYPE
FUNCTION = <string>function_name
#
# additional command
SCALE FACTOR = <real>scale_factor(1.0)
#
# Specialized initialization method
CALCULATE AS CLOSEST PROJECTION TO SURFACE =
    <string list>closeSurfs

END [INITIAL CONDITION]

```

Sierra/SM supports a general initialization procedure for setting the value of any variable. This procedure can be used to set material state variables, shell thickness, initial stress, etc. The initialization is performed both before and after the element and material model initialization. This allows the elements and material models to compute other initial variables based on variables specified by the user and also ensures that the variables specified by the user are not overwritten by the elements and material models. However, there is minimal checking in Sierra/SM to ensure that the changes made yield a consistent system. There is also no guarantee that the changes will not be overwritten or misinterpreted by some other internal routine depending on what variable is being changed. Thus, caution is advised when using this capability.

The `INITIAL CONDITION` command block, which appears in the region scope, is used to select a method and set values for initializing a variable. The command block specifies the initial value of a global variable or a variable associated with a set of mesh entities (nodes, edges, faces, or elements). The user has three options for setting initial values: with a constant magnitude, with an input mesh variable, or by a user subroutine. Only one of these three options can be specified in the command block.

The command block contains five groups of commands—mesh-entity set, variable identification, magnitude, input mesh variable, and user subroutine. In addition to the command lines in the five groups, there is one additional command line: `SCALE FACTOR`. Following are descriptions of the different command groups and the `SCALE FACTOR` command line.

7.3.1 Mesh-Entity Set Commands

The mesh-entity set commands portion of the `INITIAL CONDITION` command block specifies the nodes, element faces, or elements associated with the variable to be initialized. This portion of the command block can include some combination of the following command lines:

```

NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS

```

```

REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes, element faces, or elements. See Section 7.1.1 for more information about the use of these command lines for mesh entities. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.3.2 Variable Identification Commands

Any variable used in the `INITIAL CONDITION` command block must exist in Sierra/SM. The variable can be any currently defined variable in Sierra/SM or any user-defined variable created with the `USER VARIABLE` command block (see Section 11.2.4).

There are two command lines that identify the variable:

```

INITIALIZE VARIABLE NAME = <string>var_name
VARIABLE TYPE = [NODE|EDGE|FACE|ELEMENT|GLOBAL]

```

The `INITIALIZE VARIABLE NAME` command line gives the name of the variable for which initial values are being assigned. The variable name must be a name known by Sierra/SM. The variable name may use component syntax to initialize only specific integration points or components of the variable, see Section 9.1 for more information on component syntax.

The `VARIABLE TYPE` command line provides additional information about the variable being initialized. The options `NODE`, `EDGE`, `FACE`, `ELEMENT`, and `GLOBAL` on the command line indicate whether the variable is, respectively, a nodal, edge, face, element, or global quantity. One of these options must appear in the `VARIABLE TYPE` command line. The `GLOBAL` variable type cannot be read from an external mesh database (see Section 7.1.3).

Both of these command lines are required regardless of the option selected to set values for the variable.

7.3.3 Specification Command

If the constant magnitude command is used, one or more initial values are specified directly in the command block. This is done using the following command line:

```

MAGNITUDE = <real list>initial_values

```

The `initial_values` specified on the `MAGNITUDE` command line will set the values for the variable given by `var_name` in the `INITIALIZE VARIABLE NAME` command line. The number of values is dependent on the type of the variable specified in the `INITIALIZE VARIABLE NAME` command line. For example, if the user wanted to initialize the velocity at a set of nodes, three quantities would have to be specified since the velocity at a node is a vector quantity. If the user wanted to initialize the stress tensor for a set of uniform-gradient, eight-node hexahedral elements, six quantities would have to be specified since the stress tensor for this element type is described with six values.

7.3.4 Weibull Probability Distribution Commands

The field to be initialized can optionally be initialized or scaled with random numbers generated to conform to a specified Weibull probability distribution function. This is accomplished by including some of the following commands:

```
WEIBULL SHAPE = <real>shapeVal
WEIBULL SCALE = <real>scaleVal
WEIBULL MEDIAN = <real>medianVal
WEIBULL SEED = <int>seedVal (123456)
```

The Weibull commands may be used in conjunction with the `MAGNITUDE` command or the `READ VARIABLE` command to set the initial value, m_i , that will be scaled by the distribution. Otherwise m_i starts off equal to 1.0.

Two and only two of the parameters `WEIBULL SHAPE`, `WEIBULL SCALE`, and `WEIBULL MEDIAN` may be specified. The relationship among `WEIBULL SHAPE`, k , (also known as the Weibull modulus) `WEIBULL SCALE`, λ , and `WEIBULL MEDIAN`, μ , is shown in the following equation:

$$\mu = \lambda(\ln(2))^{\frac{1}{k}} \quad (7.1)$$

The `WEIBULL SEED` parameter sets the starting point for the random number generation. Be careful when initializing two independent fields with Weibull distributions: if the default seed (or the same user specified seed) is used for both field, the random numbers generated will be the same for both fields and the fields will therefore be completely correlated element by element.

The values at each point can be further scaled by another factor, α_i . If no scaling commands are used then $\alpha_i = 1.0$. Scaling is accomplished by using the Weibull scaling commands:

```
WEIBULL SCALING FIELD TYPE = NODE|EDGE|FACE|ELEMENT|
GLOBAL(ELEMENT)
WEIBULL SCALING FIELD NAME = <string>fieldVal (VOLUME)
WEIBULL SCALING REFERENCE VALUE = <real>value
WEIBULL SCALING EXPONENT SCALE = <real>scaleVal (1.0)
```

The scaling commands are implemented to allow the use of any field variable for scaling; however, the field type must match the type of field being set. The `WEIBULL SCALING FIELD NAME` gives the name of a variable and the `WEIBULL SCALING FIELD TYPE` gives the variable field type which can be one of nodal, edge, face, element, or global type. The `WEIBULL SCALING FIELD TYPE` and `WEIBULL SCALING FIELD NAME` default to `ELEMENT` and `VOLUME` respectively. The `WEIBULL SCALING REFERENCE VALUE` gives the reference value v_{ref} . A `WEIBULL SCALING EXPONENT SCALE`, β , may also be defined and will default to 1.0 if not defined. The scaling factor α_i is formed with the following formula:

$$\alpha_i = \left(\frac{v_{ref}}{v_i} \right)^{\frac{\beta}{k}} \quad (7.2)$$

where v_i is the value of a field variable specified with WEIBULL SCALING FIELD TYPE and WEIBULL SCALING FIELD NAME at the current point i , and the rest of the variables are those defined previously.

The scaled initial value is determined by the equation:

$$InitialValue = m_i \alpha_i \mu \left(\frac{\ln(R)}{\ln(\frac{1}{2})} \right)^{\frac{1}{k}} \quad (7.3)$$

where R is a random number in an uniform distribution between 0 and 1.

7.3.5 External Mesh Database Commands

```
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
```

See Section 7.1.3 for more details.

If the external database option is used, the initial values for a variable are read from an external mesh database. As an example, suppose the mesh file contains a set of nodal temperatures. These temperature values (which can vary for each node) can be used to initialize a temperature value associated with each node.

The values are read from a finite element model defined via the `FINITE ELEMENT MODEL`. The `READ VARIABLE = <string>var_name` command is used to read the variable from the region's finite element mesh database. The `var_name` string specifies the name of the variable as it appears on the mesh database. The number of values associated with the variable in the mesh file must be the same number associated with the variable name specified in the `INITIALIZE VARIABLE NAME` command line. For example, if `read_var_name` refers to a vector variable stored in the region's finite element mesh database and `init_var_name` refers to a vector variable being initialized, then the following command only works if both vectors have the same number of components:

```
INITIALIZE VARIABLE NAME = <string>init_var_name
READ VARIABLE = <string>read_var_name
```

Alternatively, individual components of a variable can be assigned using `<string>var_name(<int>component)`. For example, the following command line works for reading the first component of a vector and assigning it to the first component of an initialized vector:

```
INITIALIZE VARIABLE NAME = <string>init_var_name(1)
READ VARIABLE = <string>read_var_name(1)
```

The variable name used on the mesh file can be arbitrary. The name can be identical to or different from the variable name specified on the `INITIALIZE VARIABLE NAME` command line.

The field to be read may be specified at an arbitrary number of different times on the mesh file. The default behavior is to use the value of the variable at the initial time in the analysis to prescribe the initial condition. The time history is interpolated as needed for an initial analysis time that does not correspond exactly to a time on the mesh file. The `TIME` command line can optionally be used to select a specific time to initialize a variable. If the specified time on the `TIME` command line does not correspond exactly to a time on the mesh file, the data on the mesh file will be interpolated as needed. Finally, the keywords `FIRST` or `LAST` can be passed as argument to the `TIME` command in order to read values from the mesh file at the first or last time step.

7.3.6 User Subroutine Commands

If the user subroutine option is used, the initial values will be calculated by a subroutine that is written by the user explicitly for this purpose. The subroutine will be called by Sierra/SM at the appropriate time to perform the calculations.

Following are the command lines related to the user subroutine option:

```

NODE SET SUBROUTINE = <string>subroutine_name |
  SURFACE SUBROUTINE = <string>subroutine_name |
  ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
  = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
  = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
  = <string>param_value

```

Only one of the first three command lines listed above can be specified in the command block. The particular command line selected depends on the mesh-entity type of the variable being initialized. For example, variables associated with nodes would be initialized if using the `NODE SET SUBROUTINE` command line, variables associated with faces if using the `SURFACE SUBROUTINE` command line, and variables associated with elements if using the `ELEMENT BLOCK SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the selected subroutine command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The application of user subroutines for variable initialization is essentially the same as the application of user subroutines in general. See Section 7.4.10 and Chapter 11 for more details on implementing the user subroutine option.

When the user subroutine option is used for variable initialization, the user subroutine is called only once. Also, when a user subroutine is being used, the returned value is the new (initial) variable value at each mesh entity, and the flags array is ignored.

7.3.7 Distance Function Commands

```
DISTANCE TYPE = XTYPE|YTYPE|ZTYPE|RTYPE|  
                RXTYPE|RYTYPE|RZTYPE  
FUNCTION = <string>function_name
```

Usage of the `DISTANCE TYPE` command line in the `INITIAL CONDITION` command block indicates that initial values will be assigned based on the positions of the mesh entities in relation to the origin or the axes of the global coordinate system. A `FUNCTION` command line must also be given in the same `INITIAL CONDITION` block as the `DISTANCE TYPE` command line. The given function must be an xy-function, where the independent x-value is a distance quantity and the dependent y-value is returned as the initial condition value. The xy-function is evaluated at each mesh entity. The `XTYPE`, `YTYPE`, and `ZTYPE` options quantify distance as the signed x-, y-, or z-coordinate, respectively, of each mesh entity. The `RTYPE` option quantifies distance as the unsigned radial distance from the origin to the mesh entity. The `RXTYPE`, `RYTYPE`, and `RZTYPE` options quantify distance as the unsigned radial distance from the global x-axis, y-axis, and z-axis, respectively, to the mesh entity. The distance function option is only implemented for `VARIABLE TYPE = ELEMENT` and the distance quantity for each element is evaluated at the averaged nodal coordinates (approximately the centroid) of the element.

7.3.8 Additional Command

This command line provides an additional option for the `INITIAL CONDITION` command block:

```
SCALE FACTOR = <real>scale_factor(1.0)
```

Any initial value can be scaled by use of the `SCALE FACTOR` command line. An initial value generated by any one of the three initial-value-setting options in this command block (i.e., constant magnitude, input mesh, or user subroutine) will be scaled by the real value `scale_factor`.

7.3.9 Specialized Initialization Methods

A handful of highly specialized initial condition routine options are documented below.

The `CALCULATE AS CLOSEST PROJECTION TO SURFACE` initializes a three component vector at an element centroid to the direction pointing to the closest point on the given provided sidesets `closeSurfs`. Such a direction is often used for defining body fitted coordinate systems. for example a sideset could be placed on the top of a bolt head. Then this point direction direction would define the bolt aligned length direction for use in artificial strain or orthotropic material definition.

7.3.10 Examples

Initialize the stress tensors in an element block

```
# Initialize the material stress tensor on block 1 such that  
#   s_xx = s_yy = s_zz = 2.3
```

```

#   s_xy = s_xz = s_yz = 0.0
BEGIN INITIAL CONDITION
  BLOCK = block_1
  INITIALIZE VARIABLE NAME = unrotated_stress
  VARIABLE TYPE = element
  MAGNITUDE = 2.3, 2.3, 2.3, 0.0, 0.0, 0.0
END
#
# Initialize the ZZ component of the material stress tensor on
# block_2 to 1.5
#
BEGIN INITIAL CONDITION
  BLOCK = block_2
  INITIALIZE VARIABLE NAME = unrotated_stress(zz)
  VARIABLE TYPE = element
  MAGNITUDE = 1.5
END
#
# Initialize the bolt aligned coordinate system for later
# use in defining artificial preload strain
#
BEGIN USER VARIABLE pdir
  TYPE = element vector
END
BEGIN INITIAL CONDITION
  BLOCK = block_3
  INITIALIZE VARIABLE NAME = pdir
  VARIABLE TYPE = element
  CALCULATE AS CLOSEST PROJECTION TO SURFACE = sideset_3
END
BEGIN ARTIFICIAL STRAIN
  BLOCK = block_3
  DIRECTION FIELD pdir FUNCTION = shrinkFunc
END

```

7.4 Kinematic Boundary Conditions

The various kinematic boundary conditions available in Sierra/SM are described in this section. The kinematic boundary conditions are nested inside the region scope.

Kinematic constraints can potentially be in conflict with other constraints. Refer to [Appendix D](#) for information on how conflicting constraints are handled.

7.4.1 Fixed Displacement

```
BEGIN FIXED DISPLACEMENT
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# direction commands
COMPONENT = <string>X|Y|Z | COMPONENTS = <string>X/Y/Z |
  DIRECTION = <string>dirName
#
# additional commands
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [FIXED DISPLACEMENT]
```

The `FIXED DISPLACEMENT` command block fixes displacement components (X, Y, Z, or some combination thereof, or some other direction) for a set of nodes to be zero. This command block contains two groups of commands—node set and component. Each of these command groups is basically independent of the other. In addition to the command lines in the two command groups, there are two additional command lines: `ACTIVE PERIODS` and `INACTIVE PERIODS`. These are used to activate or deactivate this kinematic boundary condition for certain time periods.

For rigid bodies, any kinematic boundary condition is applied directly to the rigid body reference node. The kinematics for the nodes attached to the rigid body are applied via the rigid body constraint conditions. Since rigid bodies are defined using element blocks, only the `BLOCK` or `RIGID BODY` commands may be used to specify fixed displacement conditions on rigid bodies.



Warning: Note the fixed displacement command sets the displacement of a node to ZERO. To instead keep the current nodal displacement, a zero velocity boundary condition should be used.

Following are descriptions of the different command groups.

7.4.1.1 Node Set Commands

The `node set` commands portion of the `FIXED DISPLACEMENT` command block specifies the nodes associated with the boundary condition. This portion of the command block can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.4.1.2 Specification Commands

There are two component specification commands available in the `FIXED DISPLACEMENT` command block:

```
COMPONENT = X|Y|Z | COMPONENTS = X/Y/Z |
DIRECTION = <string>dirName
```

A single displacement component may be set with the `COMPONENT` command line. A single direction of constraint can be set with the `DIRECTION` command. Multiple components can be constrained simultaneously with the `COMPONENTS`. One and only one `COMPONENT`, `DIRECTION`, or `COMPONENTS` command must be specified in the block. The `COMPONENTS` command can constraint multiple components simultaneously, for example

```
COMPONENTS = X Y
```

7.4.1.3 Additional Commands

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines can optionally appear in the `FIXED DISPLACEMENT` command block:

```
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

These command lines determine when the boundary condition is active. See Section 2.6 for more information about these optional command lines.



Warning: If `FIXED DISPLACEMENT` becomes active the displacement is returned to zero. If the intention is to maintain the current displacement when activated `PRESCRIBED VELOCITY` with a value of zero should be used.

7.4.2 Prescribed Displacement

```
BEGIN PRESCRIBED DISPLACEMENT
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
DIRECTION = <string>defined_direction |
  COMPONENT = <string>X|Y|Z | COMPONENTS = <string>X/Y/Z |
  CYLINDRICAL AXIS = <string>defined_axis |
  RADIAL AXIS = <string>defined_axis
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
  = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
  = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
  = <string>param_value
#
# external database commands
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
TIME = <real>time|FIRST|LAST
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
```

```

    INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED DISPLACEMENT]

```

The `PRESCRIBED DISPLACEMENT` command block prescribes a displacement field for a given set of nodes. The displacement field associates a vector giving the magnitude and direction of the displacement with each node in the set of nodes. The displacement field may vary over time and space. If the displacement field has only a time-varying magnitude and uses one of four methods for setting direction, the specification commands in the above command block can be used to specify the displacement field. If the displacement field is more complex, a user subroutine is used to specify the displacement field. The displacement field can also be read from an external mesh database. In a given boundary condition command block, commands from only one of the command groups (specification commands, user subroutine commands, or external database commands) may be used.

For rigid bodies, any kinematic boundary condition is applied directly to the rigid body reference node. The kinematics for the mesh nodes attached to the rigid body are applied via the rigid body constraint conditions. Since rigid bodies are defined using element blocks, only the `BLOCK` or `RIGID BODY` command lines may be used to specify prescribed displacement conditions on rigid bodies.

The `PRESCRIBED DISPLACEMENT` command block contains four groups of commands—node set, function, user subroutine, and external database. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with the specification commands, the user subroutine commands, or the external database command. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this kinematic boundary condition for certain time periods. Following are descriptions of the different command groups and the `SCALE FACTOR` and `ACTIVE PERIODS` command lines.

7.4.2.1 Node Set Commands

The `node set` commands portion of the `PRESCRIBED DISPLACEMENT` command block defines a set of nodes associated with the prescribed displacement field and can include some combination of the following command lines:

```

NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating

a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.4.2.2 Specification Commands

If the specification commands are used, the displacement vector at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED DISPLACEMENT` command block.

Following are the command lines used to specify the prescribed displacement with a direction and a function:

```
DIRECTION = <string>defined_direction |  
  COMPONENT = <string>X|Y|Z |  
  CYLINDRICAL AXIS = <string>defined_axis |  
  RADIAL AXIS = <string>defined_axis  
FUNCTION = <string>function_name
```

The displacement can be specified along an arbitrary user-defined direction, along a component direction (X, Y, or Z), along the azimuthal direction in a cylindrical coordinate system (defined in reference to an axis), or along a radial direction (defined in reference to an axis). Only one of these options is allowed per command block. The displacement is prescribed only in the specified direction. A prescribed displacement boundary condition does not influence the motion in directions orthogonal to the prescribed direction.

- The `DIRECTION` command line is used to prescribe displacement in an arbitrary user-defined direction. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the `SIERRA` scope.
- The `COMPONENT` command line is used to specify that the prescribed displacement vector lies along one of the component directions. The `COMPONENT` command line is a shortcut to an internally defined direction vector; for example, component `x` corresponds to using direction vector (1, 0, 0).
- The `CYLINDRICAL AXIS` command line is used to specify that the prescribed displacement is to be applied in the azimuthal direction of a cylindrical coordinate system. The string `defined_axis` refers to the name of the axis of the cylindrical coordinate system, and which is defined via a `DEFINE AXIS` command block in the `SIERRA` scope. The displacement is prescribed as a rotation in radians about the axis. Nodes with this type of boundary condition are free to move in the radial and height directions in the cylindrical coordinate system. Restraints can be placed on the node set in those directions if desired by applying separate kinematic boundary conditions that contain `RADIAL AXIS` or `DIRECTION` commands that refer to the same axis. Note that this type of boundary condition is not a rotational boundary condition; it only affects translational degrees of freedom.



Known Issue: If a prescribed displacement with the `CYLINDRICAL AXIS` option is applied to nodes that fall on the axis, it will have no effect. Separate boundary conditions should be applied to those nodes to fix them in the plane normal to the axis.

- The `RADIAL AXIS` command line requires an axis definition that appears in the SIERRA scope. The string `defined_axis` uses an `axis_name` that is defined in the SIERRA scope (via a `DEFINE AXIS` command line). For this option, a radial line is drawn from a node to the radial axis. The prescribed displacement vector lies along this radial line from the node to the radial axis.

The magnitude of the displacement is specified by the `FUNCTION` command line. This references a `function_name` (defined in the SIERRA scope using a `FUNCTION` command block) that specifies the magnitude of the displacement vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in Section 7.4.2.5.

7.4.2.3 User Subroutine Commands

If the user subroutine option is used, the displacement vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED DISPLACEMENT` command block. The user subroutine option allows for a more complex description of the displacement field than do the specification commands, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a displacement direction and a magnitude for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the displacement field.

Following are the command lines related to the user subroutine option:

```

NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value

```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.4.2.5.

See Section 7.4.10 and Chapter 11 for more details on implementing the user subroutine option.

7.4.2.4 External Mesh Database Commands

If the external database option is used, the displacement vector (or specified components of the vector) is read from an external mesh database. The displacements are read from a finite element model defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model can either be the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3), or it can be a different (but compatible) model. The following command lines control the use of an external mesh database to prescribe the displacement:

```
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
  MAP_BY_PROXIMITY|MAP_BY_ID(MAP_BY_PROXIMITY)
TIME = <real>time |FIRST|LAST
```

See Section 7.1.3 for more details.

7.4.2.5 Additional Commands

These command lines in the `PRESCRIBED DISPLACEMENT` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification commands or the user subroutine. For example, if the magnitude of the displacement in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the displacement from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.4.3 Prescribed Velocity

```
BEGIN PRESCRIBED VELOCITY
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
```

```

REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
DIRECTION = <string>defined_direction |
    COMPONENT = <string>X|Y|Z | COMPONENTS = <string>X/Y/Z |
    CYLINDRICAL AXIS = <string>defined_axis |
    RADIAL AXIS = <string>defined_axis
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#
# external database commands
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED VELOCITY]

```

The `PRESCRIBED VELOCITY` command block prescribes a velocity field for a given set of nodes. The velocity field associates a vector giving the magnitude and direction of the velocity with each node in the node set. The velocity field may vary over time and space. If the velocity field has only a time-varying magnitude and uses one of four methods for setting direction, the specification commands in the above command block can be used to specify the velocity field. If the velocity field is more complex, a user subroutine is used to specify the velocity field. The velocity field can also be read from an external mesh database. In a given boundary condition command block, commands from only one of the command groups (specification commands, user subroutine commands, or external database commands) may be used.

For rigid bodies, any kinematic boundary condition is applied directly to the rigid body reference node. The kinematics for the mesh nodes attached to the rigid body are applied via the rigid body constraint conditions. Since rigid bodies are defined using element blocks, only the `BLOCK` and `RIGID BODY` command lines may be used to specify prescribed velocity conditions on rigid bodies.

The `PRESCRIBED VELOCITY` command block contains four groups of commands—node set, function, user subroutine, and external database. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with either the specification commands or the user subroutine commands. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this kinematic boundary condition for certain time periods. Following are descriptions of the different command groups.

7.4.3.1 Node Set Commands

The `node set commands` portion of the `PRESCRIBED VELOCITY` command block defines a set of nodes associated with the prescribed velocity field and can include some combination of the following command lines:

```

NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.4.3.2 Specification Commands

If the specification commands are used, the velocity vector at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED VELOCITY` command block.

Following are the command lines used to specify the prescribed velocity with a direction and a function:

```

DIRECTION = <string>defined_direction |
COMPONENT = <string>X|Y|Z |
CYLINDRICAL AXIS = <string>defined_axis |
RADIAL AXIS = <string>defined_axis
FUNCTION = <string>function_name

```

The velocity can be specified along an arbitrary user-defined direction, along a component direction (X, Y, or Z), along the azimuthal direction in a cylindrical coordinate system (defined in reference to an axis), or along a radial direction (defined in reference to an axis). Only one of these options

is allowed per command block. The velocity is prescribed only in the specified direction. A prescribed velocity boundary condition does not influence the motion in directions orthogonal to the prescribed direction.

- The `DIRECTION` command line is used to prescribe velocity in an arbitrary user-defined direction. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the SIERRA scope.
- The `COMPONENT` command line is used to specify that the prescribed velocity vector lies along one of the component directions. The `COMPONENT` command line is a shortcut to an internally defined direction vector; for example, `component x` corresponds to using direction vector (1, 0, 0).
- The `CYLINDRICAL AXIS` command line is used to specify that the prescribed velocity is to be applied in the azimuthal direction of a cylindrical coordinate system. The string `defined_axis` refers to the name of the axis of the cylindrical coordinate system, and which is defined via a `DEFINE AXIS` command block in the SIERRA scope. The velocity is prescribed as a rotation in radians about the axis. Nodes with this type of boundary condition are free to move in the radial and height directions in the cylindrical coordinate system. Restraints can be placed on the node set in those directions if desired by applying separate kinematic boundary conditions that contain `RADIAL AXIS` or `DIRECTION` commands that refer to the same axis. Note that this type of boundary condition is not a rotational boundary condition; it only affects translational degrees of freedom.



Known Issue: If a prescribed velocity with the `CYLINDRICAL AXIS` option is applied to nodes that fall on the axis, it will have no effect. Separate boundary conditions should be applied to those nodes to fix them in the plane normal to the axis.

- The `RADIAL AXIS` command line requires an axis definition that appears in the SIERRA scope. The string `defined_axis` uses an `axis_name` that is defined in the SIERRA scope (via a `DEFINE AXIS` command line). For this option, a radial line is drawn from a node to the radial axis. The velocity vector lies along this radial line from the node to the radial axis.

The magnitude of the velocity is specified by the `FUNCTION` command line. This references a `function_name` (defined in the SIERRA scope using a `FUNCTION` command block) that specifies the magnitude of the velocity vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in [Section 7.4.3.5](#).

7.4.3.3 User Subroutine Commands

If the user subroutine option is used, the velocity vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED VELOCITY` command block. The user subroutine option allows for a more complex description of the velocity field than do the specification commands, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a velocity direction and a magnitude for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the velocity field.

Following are the command lines related to the user subroutine option:

```
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.4.3.5.

7.4.3.4 External Mesh Database Commands

If the external database option is used, the velocity vector (or specified components of the vector) is read from an external mesh database. The velocities are read from a finite element model defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model can either be the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3), or it can be a different (but compatible) model. The following command lines control the use of an external mesh database to prescribe the velocity:

```
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID(MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
```

See Section 7.1.3 for more details.

7.4.3.5 Additional Commands

These command lines in the `PRESCRIBED VELOCITY` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification commands or the user subroutine. For example, if the magnitude of the velocity in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the velocity from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.4.4 Prescribed Acceleration

```
BEGIN PRESCRIBED ACCELERATION
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
DIRECTION = <string>defined_direction |
  COMPONENT = <string>X|Y|Z | COMPONENTS = <string>X/Y/Z |
  CYLINDRICAL AXIS = <string>defined_axis |
  RADIAL AXIS = <string>defined_axis
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
  = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
  = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
  = <string>param_value
#
# external database commands
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
  MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
```



```

ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED ACCELERATION]

```

The `PRESCRIBED ACCELERATION` command block prescribes an acceleration field for a given set of nodes. The acceleration field associates a vector giving the magnitude and direction of the acceleration with each node in the node set. The acceleration field may vary over time and space. If the acceleration field has only a time-varying component, the specification commands in the above command block can be used to specify the acceleration field. If the acceleration field has both time-varying and spatially varying components, a user subroutine is used to specify the acceleration field. The acceleration field can also be read from an external mesh database. In a given boundary condition command block, commands from only one of the command groups (specification commands, user subroutine commands, or external database commands) may be used.

For rigid bodies, any kinematic boundary condition is applied directly to the rigid body reference node. The kinematics for the mesh nodes attached to the rigid body are applied via the rigid body constraint conditions. Since rigid bodies are defined using element blocks, only the `BLOCK` and `RIGID BODY` commands may be used to specify prescribed acceleration conditions on rigid bodies.

The `PRESCRIBED ACCELERATION` command block contains four groups of commands: node set, function, user subroutine, and external database. Each of these command groups is independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with either the specification commands or the user subroutine commands. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this kinematic boundary condition for certain time periods. Following are descriptions of the different command groups.

7.4.4.1 Node Set Commands

The `node set` commands portion of the `PRESCRIBED ACCELERATION` command block defines a set of nodes associated with the prescribed acceleration field and can include some combination of the following command lines:

```

NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating

a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.4.4.2 Specification Commands

If the specification commands are used, the acceleration vector at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED ACCELERATION` command block. The direction of the acceleration vector is constant for all time; the magnitude of the acceleration vector may vary with time, however.

Following are the command lines used to specify the prescribed acceleration with a direction and a function:

```
DIRECTION = <string>defined_direction |  
  COMPONENT = <string>X|Y|Z  
  CYLINDRICAL AXIS = <string>defined_axis |  
  RADIAL AXIS = <string>defined_axis  
FUNCTION = <string>function_name
```

The acceleration can be specified along an arbitrary user-defined direction, along a component direction (X, Y, or Z), along the azimuthal direction in a cylindrical coordinate system (defined in reference to an axis), or along a radial direction (defined in reference to an axis). Only one of these options is allowed per command block. The acceleration is prescribed only in the specified direction. A prescribed acceleration boundary condition does not influence the motion in directions orthogonal to the prescribed direction.

- The `DIRECTION` command line is used to prescribe acceleration in an arbitrary user-defined direction. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the `SIERRA` scope.
- The `COMPONENT` command line is used to specify that the prescribed acceleration vector lies along one of the component directions. The `COMPONENT` command line is a shortcut to an internally defined direction vector; for example, component `x` corresponds to using direction vector (1, 0, 0).

The magnitude of the acceleration is specified by the `FUNCTION` command line. This references a `function_name` (defined in the `SIERRA` scope using a `FUNCTION` command block) that specifies the magnitude of the acceleration vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in [Section 7.4.4.5](#).

7.4.4.3 User Subroutine Commands

If the user subroutine option is used, the acceleration vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED ACCELERATION` command block. The user subroutine option allows for a more complex description of the acceleration field than do the specification commands, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define an acceleration direction and a magnitude for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the acceleration field.

Following are the command lines related to the user subroutine option:

```

NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value

```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.4.4.5.

See Section 7.4.10 and Chapter 11 for more details on implementing the user subroutine option.

7.4.4.4 External Mesh Database Commands

If the external database option is used, the acceleration vector (or specified components of the vector) is read from an external mesh database. The accelerations are read from a finite element model defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model can either be the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3), or it can be a different (but compatible) model. The following command lines control the use of an external mesh database to prescribe the acceleration:

```

READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST

```

See Section 7.1.3 for more details.

7.4.4.5 Additional Commands

These command lines in the `PRESCRIBED ACCELERATION` command block provide additional options for the boundary condition:

```

SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names

```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification commands or the user subroutine. For example, if the magnitude of the acceleration in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the acceleration from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.4.5 Periodic Boundary Condition

```

BEGIN PERIODIC
  SLAVE   = <string>slave_node_set
  MASTER = <string>master_node_or_face_set
  COMPONENT = <string>X/Y/Z
  PRESCRIBED QUANTITY = DISPLACEMENT|FORCE|VELOCITY
  SEARCH TOLERANCE = <real>tol
  FUNCTION = <string>func_name
  SCALE FACTOR = <real>func_scale
  POINT ON AXIS = <string>point_name
  REFERENCE AXIS = <string>direction_name
  THETA = <real>theta
  ACTIVE PERIODS = <string list>periods_names
  INACTIVE PERIODS = <string list>periods_names
END [PERIODIC]

```

The `PERIODIC` boundary condition command is used to define a set of periodic boundary constraints between two surfaces.

The `slave_node_set` is the name of the slave node set for the constraints. The `master_node_or_face_set` is the name of either the master node set or master surface for the constraint. If a master node set is used the master node set and slave node set must have the same number of nodes. Additionally the master node set and slave node set must be related by a simple translation. When two node sets are specified, a periodic constraint is defined between each pair of coincident (after applying the translation) nodes in the two node sets.

When a node set and a surface are specified the constraints will be defined between the slave nodes and the faces of the master surface. The slave node set and master face set must still be related to one another by a simple translation, but the number of nodes and mesh facets on the surfaces do not need to match. Better results are obtained when the more finely meshed surface is used as the slave node set and the coarser surface is used as the master surface.

`COMPONENT` specifies which directional component should enforce the prescribed quantity. This quantity can be `DISPLACEMENT`, `VELOCITY`, or `FORCE` as defined by the `PRESCRIBED QUANTITY`

command. Note, all degrees of freedom on the node are considered periodic. Only the `COMPONENT` will have the `PRESCRIBED QUANTITY` enforced, the other degrees of freedom will be effectively linked with zero offset unless overridden by another periodic boundary condition.

Note that neither of the `COMPONENT` or `PRESCRIBED QUANTITY` command are needed when dealing with rotational symmetry.

The `FUNCTION` and `SCALE FACTOR` commands are used to define a relative difference in the prescribed quantity on the slave and master sets. This function can be used to define an isotropic expansion or contraction of a periodic boundary. For the common default use case the function is set to zero meaning the prescribed quantities on the two sets are the same.

The `SEARCH TOLERANCE` command specifies a search tolerance used for matching up nodes and faces between the master and slave sets. The search tolerance is necessary to account for slightly varying mesh topologies between the master and slave sets. The search tolerance should be something small, generally some small fraction of the characteristic face length of facets on the surfaces.

The commands `POINT ON AXIS`, `REFERENCE AXIS`, and `THETA` are used to define axis-symmetric periodic wedge conditions. `point_name` gives the name of a point defined by a domain level `DEFINE POINT` command. `direction_name` gives the name of a point defined by a domain level `DEFINE DIRECTION` command. Taken together these two commands define the axis of rotation for the wedge body. `THETA` defines the angle, in degrees, covered by the wedge. By way of the thumb rule, `THETA` should always be positive when sweeping from master to slave, in a positive rotation.



Warning: The axis-symmetric periodic boundary condition capability is still in development and has only been evaluated for implicit, serial analysis.

Examples:

```
SLAVE           = nodeset_11
MASTER          = nodeset_12
COMPONENT       = xz
PRESCRIBED QUANTITY = displacement
SEARCH TOLERANCE = 0.001
FUNCTION        = function_1
SCALE FACTOR    = 0.0
```

The above commands will create a periodic boundary condition between node sets 11 and 12. The translational offset between `nodeset_11` and `nodeset_12` will be automatically determined. The coincident (after the translation is applied) nodes in these node sets will be matched up. For each node pair a constraint will be created to ensure the x and z displacements of the two nodes in the pair are equal.



Warning: Periodic boundary conditions should generally not be combined with other kinematic boundary conditions (such as fixed displacement.) The periodic boundary conditions define a full set of kinematic constraints and additional constraints may conflict with the periodic constraints in unexpected ways.

7.4.6 Fixed Rotation

```
BEGIN FIXED ROTATION
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
COMPONENT = <string>X/Y/Z | COMPONENTS = <string>X/Y/Z |
  DIRECTION = <string>dir_name
#
# additional commands
ACTIVE PERIODS = <string list>periods_names
INACTIVE PERIODS = <string list>periods_names
END [FIXED ROTATION]
```

The `FIXED ROTATION` command is only applied to nodes that have rotational degrees of freedom such as shell element nodes and rigid body reference nodes. In the case of rigid bodies, the boundary condition must be specified on the block that defines the rigid body using the `BLOCK` or `RIGID BODY` line command.

The `FIXED ROTATION` command block fixes rotation about direction components (X, Y, Z, or some combination thereof) for a set of nodes to zero. This command block contains two groups of commands—node set and component. Each of these command groups is basically independent of the other. In addition to the command lines in the two command groups, there are additional command lines: `ACTIVE PERIODS` and `INACTIVE PERIODS`. These command lines are used to activate or deactivate this kinematic boundary condition for certain time periods.



Warning: Note the fixed rotation command sets the rotation of a node to ZERO. To instead keep the current nodal displacement, a zero rotational velocity boundary condition should be used.

Following are descriptions of the different command groups.

7.4.6.1 Node Set Commands

The `node set commands` portion of the command block specifies the nodes associated with the boundary condition. This portion of the command block can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.4.6.2 Specification Commands

There are two component specification commands available in the `FIXED ROTATION` command block:

```
COMPONENT = X|Y|Z | COMPONENTS = X/Y/Z |
DIRECTION = <string>dirName
```

A single rotation component may be set with the `COMPONENT` command line. A single direction of constraint can be set with the `DIRECTION` command. Multiple components can be constrained simultaneously with the `COMPONENTS`. One and only one `COMPONENT`, `DIRECTION`, or `COMPONENTS` command must be specified in the block. The `COMPONENTS` command can constraint multiple components simultaneously, for example

```
COMPONENTS = X Y
```

7.4.6.3 Additional Commands

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines can optionally appear in the `FIXED ROTATION` command block:

```
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

This command line determines when the boundary condition is active. See Section 2.6 for more information about this optional command line.

7.4.7 Prescribed Rotation

```
BEGIN PRESCRIBED ROTATION
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
DIRECTION = <string>defined_direction
COMPONENT = X|Y|Z
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#
# external database commands
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED ROTATION]
```



Warning: The BEGIN PRESCRIBED ROTATION command is only applied to nodes that have rotational degrees of freedom such as shell element nodes and rigid body reference nodes. Displacements consistent with rotation about a fixed axis for nodes that do not have rotational degrees of freedom, are imposed using the CYLINDRICAL AXIS command line in the PRESCRIBED DISPLACEMENT command block described in Section 7.4.2.



Warning: The behavior of the `BEGIN PRESCRIBED ROTATION` command differs for implicit and explicit analyses. For implicit analyses, the `BEGIN PRESCRIBED ROTATION` command fully prescribes all rotational degrees of freedom. Consequently, a prescribed rotation in a given direction restricts rotation in the remaining two orthogonal directions. If multiple prescribed rotation boundary conditions are applied to a node, only the last one specified in the input is enforced.

In explicit analyses, the `BEGIN PRESCRIBED ROTATION` command block prescribes rotational degree of freedom in only the specified direction. The node is free to rotate in the directions orthogonal to that direction. Multiple prescribed rotation boundary conditions can be applied to constrain rotations in those other directions if desired.

For nodal rotational degrees of freedom, the rotations applied using the `BEGIN PRESCRIBED ROTATION` command are the three components of the rotational degree of freedom itself. For rigid bodies, the rotations must be the nonlinear components that result in the desired quaternion on the rigid body reference node. For rigid bodies, the boundary condition is applied to the reference node and therefore must be specified on the block that defines the rigid body using the `BLOCK` or `RIGID BODY` line command.

The `PRESCRIBED ROTATION` command block prescribes the rotation about an axis for a given set of nodes. The rotation field associates a vector giving the magnitude and direction of the rotation with each node in the node set. The rotation field may vary over time and space. If the rotation field has only a time-varying component, the specification commands in the above command block can be used to specify the rotation field. If the rotation field has both time-varying and spatially varying components, a user subroutine is used to specify the rotation field. The rotation field can also be read from an external mesh database. In a given boundary condition command block, commands from only one of the command groups (specification commands, user subroutine commands, or external database commands) may be used.

The `PRESCRIBED ROTATION` command block contains four groups of commands—node set, function, user subroutine, and external database. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with either the specification commands or the user subroutine commands. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this kinematic boundary condition for certain time periods. Following are descriptions of the different command groups.

7.4.7.1 Node Set Commands

The `node set commands` portion of the `PRESCRIBED ROTATION` command block defines a set of nodes associated with the prescribed rotation field and can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.4.7.2 Specification Commands

If the specification commands are used, the rotation vector at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED ROTATION` command block. The direction of the rotation vector is constant for all time; the magnitude of the rotation vector may vary with time, however.

Following are the command lines used to specify the prescribed rotation with a direction and a function:

```
DIRECTION = <string>defined_direction
COMPONENT = X|Y|Z
FUNCTION = <string>function_name
```

The `DIRECTION` or `COMPONENT` command line are used to prescribe the direction of rotation. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the `SIERRA` scope. The rotation is prescribed only in the specified direction. A prescribed rotation boundary condition does not influence the rotation in directions orthogonal to the prescribed direction.

The magnitude of the rotation is specified by the `FUNCTION` command line. This references a `function_name` (defined in the `SIERRA` scope in a `FUNCTION` command block) that specifies the magnitude of the rotation vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in Section 7.4.7.5.

The magnitude of the rotation, as specified by the product of the function and the scale factor, has units of radians per second.

7.4.7.3 User Subroutine Commands

If the user subroutine option is used, the rotation vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED ROTATION` command block. The user subroutine option allows for a more complex description of the rotation field than do the specification commands, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a rotation direction and a magnitude for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the rotation field.

Following are the command lines related to the user subroutine option:

```
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.4.7.5.

See Section 7.4.10 and Chapter 11 for more details on implementing the user subroutine option.

7.4.7.4 External Mesh Database Commands

If the external database option is used, the rotation vector (or specified components of the vector) is read from an external mesh database. The rotations are read from a finite element model defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model can either be the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3), or it can be a different (but compatible) model. The following command lines control the use of an external mesh database to prescribe the rotation:

```
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
```

See Section 7.1.3 for more details.

7.4.7.5 Additional Commands

These command lines in the `PRESCRIBED ROTATION` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification commands or the user subroutine. For example, if the magnitude of the rotation in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the rotation from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.4.8 Prescribed Rotational Velocity

```
BEGIN PRESCRIBED ROTATIONAL VELOCITY
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
DIRECTION = <string>defined_direction |
  COMPONENT = <string>X|Y|Z
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
  = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
  = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
  = <string>param_value
```

```

#
# external database commands
READ VARIABLE = <string>variable_name
COPY VARIABLE = <string>variable_name [FROM MODEL
    <string>model_name]
TIME = <real>time|FIRST|LAST
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED ROTATIONAL VELOCITY]

```



Warning: The `PRESCRIBED ROTATIONAL VELOCITY` command is only applied to nodes that have rotational degrees of freedom such as shell element nodes and rigid body reference nodes. Velocities consistent with rotation about a fixed axis for nodes that do not have rotational degrees of freedom, are imposed with the `CYLINDRICAL AXIS` command line in the `PRESCRIBED VELOCITY` command block described in Section [7.4.3](#).



Warning: The behavior of the `BEGIN PRESCRIBED ROTATIONAL VELOCITY` command differs for implicit and explicit analyses. For implicit analyses, the `BEGIN PRESCRIBED ROTATIONAL VELOCITY` command fully prescribes all rotational degrees of freedom. Consequently, a prescribed rotational velocity in a given direction restricts rotation in the remaining two orthogonal directions. If multiple prescribed rotational velocity boundary conditions are applied to a node, only the last one specified in the input is enforced.

In explicit analyses, the `BEGIN PRESCRIBED ROTATIONAL VELOCITY` command block prescribes rotational degree of freedom in only the specified direction. The node is free to rotate in the directions orthogonal to that direction. Multiple prescribed rotational velocity boundary conditions can be applied to constrain rotations in those other directions if desired.

The `PRESCRIBED ROTATIONAL VELOCITY` command block prescribes the rotational velocity about an axis for a given set of nodes. The rotational velocity field associates a vector giving the magnitude and direction of the rotational velocity with each node in the node set. The rotational velocity field may vary over time and space. If the rotational velocity field has only a time-varying component, the specification commands in the above command block can be used to specify the rotational velocity field. If the rotational velocity field has both time-varying and spatially varying components, a user subroutine is used to specify the rotational velocity field. The rotational velocity field can also be read from an external mesh database. In a given boundary condition command block, commands from only one of the command groups (specification commands, user subroutine

commands, or external database commands) may be used.

For rigid bodies, the user must specify the rotational velocity components such that they define the desired quaternion on the rigid body reference node. For rigid bodies, the boundary condition is applied to the reference node and therefore must be specified on the block that defines the rigid body using the `BLOCK` or `RIGID BODY` line command.

The `PRESCRIBED ROTATIONAL VELOCITY` command block contains four groups of commands—node set, function, user subroutine, and external database. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with either the specification commands or the user subroutine commands. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this kinematic boundary condition for certain time periods. Following are descriptions of the different command groups.

7.4.8.1 Node Set Commands

The node set commands portion of the `PRESCRIBED ROTATIONAL VELOCITY` command block defines a set of nodes associated with the prescribed rotational velocity field and can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.4.8.2 Specification Commands

If the specification commands are used, the rotational velocity vector at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED ROTATIONAL VELOCITY` command block. The direction of the rotational velocity vector is constant for all time; the magnitude of the rotational velocity vector may vary with time, however.

Following are the command lines used to specify the prescribed rotational velocity with a direction and a function:

```
DIRECTION = <string>defined_direction |
COMPONENT = <string>X|Y|Z
FUNCTION = <string>function_name
```

The rotational velocity can be specified either along an arbitrary user-defined direction or along a component direction (X, Y, or Z), but not both. The rotational velocity is prescribed only in the specified direction. A prescribed rotational velocity boundary condition does not influence the rotational velocity in directions orthogonal to the prescribed direction.

- The `DIRECTION` command line is used to prescribe rotational velocity in an arbitrary user-defined direction. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the `SIERRA` scope.
- The `COMPONENT` command line is used to specify that the prescribed rotational velocity vector lies along one of the component directions. The `COMPONENT` command line is a shortcut to an internally defined direction vector; for example, `component x` corresponds to using direction vector (1, 0, 0).

The magnitude of the rotational velocity is specified by the `FUNCTION` command line. This references a `function_name` (defined in the `SIERRA` scope in a `FUNCTION` command block) that specifies the magnitude of the rotational velocity vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in Section 7.4.8.5.

The magnitude of the rotational velocity, as specified by the product of the function and the scale factor, has units of radians per second.

7.4.8.3 User Subroutine Commands

If the user subroutine option is used, the rotational velocity vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED ROTATIONAL VELOCITY` command block. The user subroutine option allows for a more complex description of the rotational velocity field than do the specification commands, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a rotational velocity direction and a magnitude for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the rotational velocity field.

Following are the command lines related to the user subroutine option:

```

NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value

```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a `FORTRAN` subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL`

PARAMETER, SUBROUTINE INTEGER PARAMETER, and SUBROUTINE STRING PARAMETER. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.4.8.5.

See Section 7.4.10 and Chapter 11 for more details on implementing the user subroutine option.

7.4.8.4 External Mesh Database Commands

If the external database option is used, the rotational velocity vector (or specified components of the vector) is read from an external mesh database. The rotational velocities are read from a finite element model defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model can either be the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3), or it can be a different (but compatible) model. The following command lines control the use of an external mesh database to prescribe the rotational velocity:

```
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
```

See Section 7.1.3 for more details.

7.4.8.5 Additional Commands

These command lines in the `PRESCRIBED ROTATIONAL VELOCITY` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification commands or the user subroutine. For example, if the magnitude of the rotational velocity in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the rotational velocity from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.4.9 Reference Axis Rotation

```
BEGIN REFERENCE AXIS ROTATION
#
```



```

# block commands
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
#
# specification commands
REFERENCE AXIS X FUNCTION = <string>function_name
REFERENCE AXIS Y FUNCTION = <string>function_name
REFERENCE AXIS Z FUNCTION = <string>function_name
#
# rotation commands
ROTATION = <string>function_name
ROTATIONAL VELOCITY = <string>function_name
#
# torque command
TORQUE = <string>function_name
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [REFERENCE AXIS ROTATION]

```

The REFERENCE AXIS ROTATION command block is intended to control the rotation of a rigid body. The three REFERENCE AXIS line commands are required and together define a time-varying reference axis or vector. The rigid body will rotate to follow the vector. Depending on other line commands in the block, the rigid body will have either zero or one free rotational degree of freedom. If one degree of freedom is present, it is the rotation about the time-varying reference vector.

At most, one of the ROTATION, ROTATIONAL VELOCITY, or TORQUE line commands may be present. If the ROTATION or ROTATIONAL VELOCITY command line is present, the rigid body will be fully prescribed for rotation. The ACTIVE PERIODS and INACTIVE PERIODS command lines are used to activate or deactivate this kinematic boundary condition for certain time periods.

Use of the REFERENCE AXIS ROTATION command block will cause extra global variables to be written to the results file. These variables describe the reaction force, rotational reaction moment, rotational displacement, and rotational velocity associated with the boundary condition and in the local coordinate system described by the boundary condition. The names of the variables are:

```

REACTR_<NAME>
REACTS_<NAME>
REACTT_<NAME>

RREACTR_<NAME>
RREACTS_<NAME>
RREACTT_<NAME>

ROTDS_<NAME>

```

```
ROTVR_<NAME>  
ROTVS_<NAME>  
ROTVT_<NAME>
```

The variables beginning with `REACT` give the reaction forces in the boundary condition's local coordinate system. Those beginning with `RREACT` give the reaction moments.

Only the `S` component is given for the rotational displacement. This is because that the other coordinate directions are poorly defined, and therefore, rotational displacements in those directions provide no value.

The variables beginning with `ROTV` give the rotational velocity in the local coordinate system.

For each global variable, `<NAME>` is the name of the rigid body controlled by this boundary condition.

7.4.9.1 Block Command

The `block` command portion of the `REFERENCE AXIS ROTATION` command block defines a block associated with the prescribed rotation field and must include the following command line:

```
BLOCK = <string list>block_names
```

If the element block in the `BLOCK` command has been defined as a rigid body, the rotation is applied to the rigid body reference node.

Alternatively, the command line:

```
RIGID BODY = <string list>rigid_body_names
```

may be used to specify the rotation on the reference node of the specified rigid body.

See Section [7.1.1](#) for more information about the use of this command line.

7.4.9.2 Specification Commands

The three specification commands are required and together define a time-varying vector that gives the orientation of a reference axis for rotation. The magnitude of the vector is ignored.

Following are the command lines used to specify the reference axis rotation with a direction:

```
REFERENCE AXIS X FUNCTION = <string>function_name  
REFERENCE AXIS Y FUNCTION = <string>function_name  
REFERENCE AXIS Z FUNCTION = <string>function_name
```

At a given time, the function referred to by the `REFERENCE AXIS X FUNCTION` line command gives the `x` component of a vector defining the reference axis. The pattern holds for the `y` and `z` components as well. Since the magnitude of the resulting axis or vector is not used, the three functions do not need to describe a unit vector in time.

7.4.9.3 Rotation Commands

It is possible to prescribe the complete rotation of a rigid body through the use of the `ROTATION` or `ROTATIONAL VELOCITY` line commands. Without these line commands, the rigid body is free to rotate about the reference axis.

Following are the command lines to prescribe the rotation about the reference axis:

```
ROTATION = <string>function_name  
ROTATIONAL VELOCITY = <string>function_name
```

The function referred to by the `ROTATION` line command gives the magnitude of rotation in radians about the reference axis as a function of time. This function should not be changed across a restarted analysis.

The function referred to by the `ROTATIONAL VELOCITY` line command gives the rotational velocity in radians per second about the reference axis as a function of time. This function should not be changed across a restarted analysis.

7.4.9.4 Torque Command

A user may specify a moment about the reference axis through the use of the `TORQUE` line command.

```
TORQUE = <string>function_name
```

The function referred to by the `TORQUE` line command gives the torque or moment about the reference axis as a function of time.

7.4.9.5 Additional Commands

The `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS` command lines can optionally appear in the `REFERENCE AXIS ROTATION` command block:

```
SCALE FACTOR = <real>scale_factor(1.0)  
ACTIVE PERIODS = <string list>period_names  
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line applies the specified value as a scale factor to the rotation, rotational velocity, or torque functions.

The final two command lines can activate or deactivate the reference axis rotation for certain time periods. See Section 2.6 for more information about these command lines.

7.4.10 Subroutine Usage for Kinematic Boundary Conditions

The prescribed kinematic boundary conditions may be defined by a user subroutine. All these conditions use a node set subroutine. See Chapter 11 for an in-depth discussion of user subroutines. The kinematic boundary conditions will be applied to nodes. The subroutine will have to return six output values per node and one output flag per node. The usage of the output values depends on the returned flag value for a node, as follows:

- If the flag value is negative, no constraint will be applied to the node.
- If the flag value is equal to zero, the constraint will be absolute. All components of the boundary condition will be specified. For example, suppose the user has specified a subroutine to be called from a prescribed displacement subroutine. The prescribed displacements are to be passed through an array `output_values`. For a given node `inode`, the `output_values` array would have the following values:

```
output_values(1,inode) = displacement in x at inode
output_values(2,inode) = displacement in y at inode
output_values(3,inode) = displacement in z at inode
output_values(4,inode) = not used
output_values(5,inode) = not used
output_values(6,inode) = not used
```

- If the flag value is equal to one, the constraint will be a specified amount in a given direction. For example, suppose the user has specified a subroutine to be called from a prescribed displacement subroutine. The prescribed displacements are to be passed through an array `output_values`. For a given node `inode`, the `output_values` array would have the following values:

```
output_values(1,inode) = magnitude of displacement
output_values(2,inode) = not used
output_values(3,inode) = not used
output_values(4,inode) = x component of direction vector
output_values(5,inode) = y component of direction vector
output_values(6,inode) = z component of direction vector
```

The direction in which the constraint will act is given by `output_values` 4 through 6 for `inode`. The magnitude of the displacement in the specified direction is given by `output_values` 1 at `inode`. To compute the constraint, Sierra/SM first normalizes the direction vector. Next, Sierra/SM multiplies the normalized direction vector by the magnitude of the displacement and applies the resultant constraint vector.

Displacements or velocities orthogonal to the prescribed direction will not be constrained. (This is true regardless of whether one uses a user subroutine for the prescribed kinematic boundary conditions.) Take the case of a prescribed displacement condition. The displacement orthogonal to a prescribed direction of motion depends on the internal and external forces orthogonal to the prescribed direction. Displacement orthogonal to the prescribed direction may or may not be zero.

7.5 Initial Velocity Conditions

```
BEGIN INITIAL VELOCITY
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# direction specification commands
COMPONENT = <string>X|Y|Z |
    DIRECTION = <string>defined_direction
#
# magnitude specification commands
MAGNITUDE = <real>magnitude_of_velocity
FUNCTION = <string>function_name
SCALE FACTOR = <real>function_scale
#
# angular velocity specification commands
CYLINDRICAL AXIS = <string>defined_axis
ANGULAR VELOCITY = <real>angular_velocity
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
END [INITIAL VELOCITY]
```

The `INITIAL VELOCITY` command block specifies an initial velocity field for a set of nodes. There are two simple options for specifying the initial velocity field: by direction and by angular velocity. The user subroutine option available is also available to specify an initial velocity. Only one of the available options—direction specification, angular velocity specification, or user subroutine—may be used.

The `INITIAL VELOCITY` command block contains four groups of commands—node set, direction specification, angular velocity specification, and user subroutine. Command lines associated with the node set commands must appear. As mentioned, command lines associated with one of the options must also appear. Following are descriptions of the different command groups.



Warning: The `INITIAL VELOCITY` command block may *not* be used to set an initial rotational velocity for rigid bodies. To do this with rigid bodies, see [6.3.1](#). This block only allows the specification of translational degrees of freedom for rigid bodies.



Warning: Initial velocity follows the same constraint enforcement hierarchy as other kinematic boundary conditions. Thus, if two non-orthogonal initial velocities are specified the result will not be an addition of the two velocities, rather the last boundary condition defined in the input deck will override the non-orthogonal components of previous boundary conditions. See [Section D](#) for further details and examples.

If it is desired to set an initial spin and translational velocity on the same part a subroutine exists in the user subroutine library to accomplish this task. See [Section 11.5.3](#).

7.5.1 Node Set Commands

The `node set commands` portion of the `INITIAL VELOCITY` command block defines a set of nodes associated with the initial velocity field and can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

For rigid bodies, the boundary condition is applied to the reference node and must be specified on the block that defines the rigid body using the `BLOCK` or `RIGID BODY` line command.

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See [Section 7.1.1](#) for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `RIGID BODY`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.5.2 Direction Specification Commands

If the direction specification commands are used, the initial velocity is applied along a defined direction with a specific magnitude. Following are the command lines for the direction option:

```
COMPONENT = <string>X|Y|Z |
```

```

DIRECTION = <string>defined_direction
MAGNITUDE = <real>magnitude_of_velocity
FUNCTION = <string>function_name
SCALE FACTOR = <real>function_scale

```

The initial velocity can be specified either along an arbitrary user-defined direction or along a component direction (X, Y, or Z), but not both. The velocity is prescribed only in the specified direction. A prescribed velocity boundary condition does not influence the movement in directions orthogonal to the prescribed direction.

- The `DIRECTION` command line is used to prescribe initial velocity in an arbitrary user-defined direction. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the `SIERRA` scope.
- The `COMPONENT` command line is used to specify that the initial velocity vector lies along one of the component directions. The `COMPONENT` command line is a shortcut to an internally defined direction vector; for example, component `x` corresponds to using direction vector (1, 0, 0).

The magnitude of the initial velocity can be given in a few ways. One option is the `MAGNITUDE` which defines a constant initial velocity for the boundary condition. A second option is use of a `FUNCTION` command. The `FUNCTION` is evaluated at the initial time to set the initial velocity. Generally the `FUNCTION` command would be used with spatially dependent analytic functions. The `SCALE FACTOR` command line defines an additional multiplier on the magnitude or magnitude function.

Either the `COMPONENT` command line or the `DIRECTION` command line must be specified when using the `MAGNITUDE` or `FUNCTION` command lines.

7.5.3 Angular Velocity Specification Commands

If the angular velocity specification commands are used, the initial velocity is applied as an initial angular velocity about some axis. Following are the command lines for angular velocity specification:

```

CYLINDRICAL AXIS = <string>defined_axis
ANGULAR VELOCITY = <real>angular_velocity

```

The axis about which the body is initially rotating is given by the `CYLINDRICAL AXIS` command line. The string `defined_axis` uses an `axis_name` that is defined in the `SIERRA` scope (via a `DEFINE AXIS` command line).

The magnitude of the angular velocity about this axis is specified by the `ANGULAR VELOCITY` command line with the real value `angular_velocity`. This value is specified in units of radians per unit of time. Typically, the value for the angular velocity will be radians per second.

Both the `CYLINDRICAL AXIS` and `ANGULAR VELOCITY` command lines are required when using the angular velocity specification commands.

7.5.4 User Subroutine Commands

If the user subroutine option is used, the initial velocity vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `INITIAL CONDITION` command block. The user subroutine option allows for a more complex description of the initial velocity field than do the direction and angular-velocity options, but the user subroutine option also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a velocity direction and a magnitude for every node to which the initial velocity field will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the initial velocity field.

Following are the command lines related to the user subroutine option:

```
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

See Section 7.4.10 and Chapter 11 for more details on implementing the user subroutine option.

7.6 Force Boundary Conditions

A variety of force boundary conditions are available in Sierra/SM. This section describes these boundary conditions.

7.6.1 Pressure

```
BEGIN PRESSURE
#
# surface set commands
SURFACE = <string list>surface_names
REMOVE SURFACE = <string list>surface_names
BLOCK = <string list> block_names
INCLUDE ALL BLOCKS
#
# specification commands
FUNCTION = <string>function_name
VELOCITY DAMPING COEFFICIENT = <real>damping_coefficient
# user subroutine commands
SURFACE SUBROUTINE = <string>subroutine_name |
    NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#
# external pressure sources
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
OBJECT TYPE = <string>NODE|FACE(NODE)

FIELD VARIABLE = <string>field_variable
SCALAR SOURCE VARIABLE = <string>var_name INDEX
    <integer>var_index REGION <string>region_name

#
# output external forces from pressure
EXTERNAL FORCE CONTRIBUTION OUTPUT NAME =
    <string>variable_name
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
```

```

    INACTIVE PERIODS = <string list>period_names

    INTEGRATION METHOD =
        CODE SELECTED | MULTILINEAR | QUADRATIC (CODE_SELECTED)

END [PRESSURE]

```

The `PRESSURE` command block applies a pressure to each face of the associated surfaces. The pressure field can be constant over the faces and vary in time, dependent on the average face velocity based on a damping coefficient, or determined by a user subroutine. If the pressure field is constant over the faces and has only a time-varying component, the function command in the above command block can be used to specify the pressure field. For a velocity-dependent pressure, a damping coefficient is specified. If the pressure field has both time-varying and spatially varying components, user subroutine commands are used to specify the pressure field. The pressure field may also be obtained from a mesh file or from another SIERRA code through a transfer operator. Only one of these options—function, velocity damping coefficient, user subroutine, mesh file, transfer from another code—may be used to specify the pressure field in a particular `PRESSURE` command block.

The `PRESSURE` command block can be used for surfaces that have faces derived from solid elements (eight-node hexahedra, four-node tetrahedra, eight-node tetrahedra, etc.), membranes, and shells. The `PRESSURE` command block can also be used for particle like elements if the elements are created through the use of element death particle conversion, Section 6.5.4.9. The surfaces must be defined on the original solid elements.

Pressure is integrated in the current deformed model configuration. Thus if due to displacement a face grows, shrinks, or rotates the net force produced by the pressure boundary condition will change.

A pressure boundary condition generates nodal forces that are summed into the external force vector that is used to calculate the motion of a body. The external force vector contains the contribution from all forces acting on the body. There is an option in the `PRESSURE` command block to save information about the contribution to the external force vector due only to pressure loads. This option does not change the magnitude or time history of the pressure load (regardless of how they are defined), but merely stores information in a user-accessible variable.



Warning: Pressure is integrated over each face and applied at the nodes with forces. The contribution to each node on a face is based on topology, which in some cases is simplified from that of the underlying element. For quadrilateral, triangular, and a few higher order faces (e.g., eight and nine node quadrilaterals) the topology used is exact, but in some cases it is an approximation formed from subdividing the face into multiple triangular faces.

The `PRESSURE` command block contains five groups of commands—surface set, function, user subroutine, external pressure, and output external forces. Each of these command groups is basically independent of the others. In addition to the command lines in the five command

groups, there are two additional command lines: `SCALE FACTOR` and `ACTIVE PERIODS`. The `SCALE FACTOR` command line will apply an additional scaling factor to the pressure value computed. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this force boundary condition for certain time periods. Following are descriptions of the different command groups.

7.6.1.1 Surface Set Commands

The `surface set` commands portion of the `PRESSURE` command block defines a set of surfaces associated with the pressure field and can include some combination of the following command lines:

```
SURFACE = <string list>surface_names
BLOCK =  <string list>block_names
INCLUDE ALL BLOCKS
REMOVE SURFACE = <string list>surface_names
```

In the `SURFACE` command line, a series of surfaces may be given through the string list `surface_names`. When using the `BLOCK` command the pressure will be applied to the exterior boundary of the block. When using the `INCLUDE ALL BLOCKS` command the pressure will be applied to the exterior boundary of all blocks in the model.

There must be at least one `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` command line in the command block. The `REMOVE SURFACE` command allows creation of an application surface formed from the union of surfaces specified in the `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` minus those surfaces specified in the `REMOVE SURFACE` command.

See Section 7.1.1 for more information about the use of these command lines for creating a set of surfaces used by the boundary condition.

7.6.1.2 Specification Commands

If the function command is used, the pressure vector at any given time is the same for all surfaces associated with the particular `PRESSURE` command block. The direction of the pressure vector is constant for all time; the magnitude of the pressure vector may vary with time.

Following is the command line used to specify the pressure with a function:

```
FUNCTION = <string>function_name
```

The pressure is applied in the opposite direction to the outward normals of the faces that define the surfaces. The magnitude of the pressure is specified by the `FUNCTION` command line. This references a `function_name` (defined in the `SIERRA` scope in a `FUNCTION` command block) that specifies the magnitude of the pressure vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in Section 7.6.1.6.

A face force based on a velocity damping coefficient can be applied using the command line:

```
VELOCITY DAMPING COEFFICIENT = <real>damping_coefficient
```

In this case, nodal forces will be applied to the nodes of any faces included in the surfaces associated with the `PRESSURE` command block. A force vector acting normal to the face is computed by multiplying the average nodal velocity normal to the face by the damping coefficient. This face force is then evenly distributed among the nodes of the face.

7.6.1.3 User Subroutine Commands

If the user subroutine option is used, the pressure may vary spatially at any given time for each of the surfaces associated with the particular `PRESSURE` command block. The user subroutine option allows for a more complex description of the pressure field than does the function command, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a pressure for every face to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the pressure field.

Following are the command lines related to the user subroutine option:

```
SURFACE SUBROUTINE = <string>subroutine_name |  
  NODE SET SUBROUTINE = <string>subroutine_name  
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON  
SUBROUTINE REAL PARAMETER: <string>param_name  
  = <real>param_value  
SUBROUTINE INTEGER PARAMETER: <string>param_name  
  = <integer>param_value  
SUBROUTINE STRING PARAMETER: <string>param_name  
  = <string>param_value
```

The user subroutine option is invoked by using the `SURFACE SUBROUTINE` command line or the `NODE SET` subroutine command line. The string `subroutine_name` in both command lines is the name of a FORTRAN subroutine that is written by the user. The particular command line selected depends on the mesh-entity type for which the pressure field is being calculated. Associating pressure values with faces would require the use of a `SURFACE SUBROUTINE` command line. Associating pressure values with nodes would require the use of a `NODE SET SUBROUTINE` command line.

Following the selected subroutine command line are other command lines that may be used to implement the user subroutine option. These command lines are described in [Section 11.2.2](#) and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout [Chapter 11](#).

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in [Section 7.6.1.6](#).

Usage requirements. Following are the usage requirements for the two types of subroutines:

- The surface subroutine operates on a group of faces. The subroutine will return one output value per face. Suppose a provided user subroutine returns the pressure information through an array `output_value`. The value `output_value(1, iface)` corresponds to the average pressure on face `iface`. The values of the flags array are not used.
- The node set subroutine provided will return one value per node. Suppose a provided user subroutine returns the pressure information through an array `output_value`. The return value `output_value(1, inode)` is the pressure at the node `inode`. The total pressure on the each face is found by integrating the pressures at the nodes. The values of the flags array are not used.

See Chapter 11 for more details on implementing the user subroutine option.

7.6.1.4 External Pressure Sources

Pressure may be obtained from two different external sources. The first option for obtaining pressure uses a mesh database. The pressures are read from a finite element model defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model can either be the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3), or it can be a different (but compatible) model. The following command lines control the use of an external mesh database to prescribe the pressure:

```

READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
OBJECT TYPE = <string>NODE|FACE (FACE)

```

See Section 7.1.3 for more details.

The `OBJECT TYPE` command line specifies whether the pressure field on the mesh file is specified for nodes (the mesh object type is `NODE`) or for faces (the mesh object type is `FACE`). If the `OBJECT TYPE` command line is not present, it is assumed that the named variable is defined on faces.

If the pressure field is defined on the nodes, the field used should have one value per node which is the node centered pressure. If the pressure field is defined on faces, there are two options for field format. The first option is to have one value defined per face, which defines a constant pressure over each face. The second option is to define *numNode* values per face where *numNode* is the number of nodes on each face (four per quad, three per triangle, etc.) When this option is used, the pressure field is defined at each corner of the face; this linearly varying pressure is then integrated over the face to compute the net forces. These nodal pressure weightings are sometimes called pressure distribution factors in other codes.

The second option for obtaining pressure from an external source relies on the transfer of information from another SIERRA code. The command for obtaining pressure information by transfer from another code is:

```
FIELD VARIABLE = <string>var_name
```

Here, `var_name` is the name of the variable where pressure information is to be stored. The pressure information will be transferred into this variable from another SIERRA code via a transfer operator.

The third option for obtaining pressure from an external source relies on the exchanging of information in a two-way coupled simulation with another SIERRA code. The command for obtaining pressure in this manner is:

```
SCALAR SOURCE VARIABLE = <string>var_name INDEX  
                          <integer>var_index REGION <string>region_name
```

In this command, `var_name` is the name of the variable where pressure information is stored in the other code. `INDEX var_index` is the index into the variable for multi-component fields. If the pressure source is a single global, the command is `INDEX 0`. `REGION region_name` is the name of the Sierra region Sierra/SM is coupling to.

An example of using this command would be the following: In the Sierra code Aria, a bulk fluid node can be used to calculate the temperature and pressure of air in an enclosed volume. Once this is calculated, the bulk fluid node pressure can then be applied to the interior surface of the container enclosing the air in the Adagio region. Similarly, this command can also be used in thermal-mechanical foam decomposition simulations.

7.6.1.5 Output Command

This command line lets the user create a variable that stores information about the contribution to the external force vector at a node arising solely from a pressure:

```
EXTERNAL FORCE CONTRIBUTION OUTPUT NAME =  
    <string>variable_name
```

If the above command line appears in a `PRESSURE` command block, then there will be a variable created with whatever name the user specifies for `variable_name`. The variable defines a three-dimensional vector at each node associated with this particular command block. The three-dimensional vector at each node represents the external force due solely to the pressure on the elements attached to that node. For example, if one of the nodes associated with this particular command block has four elements attached to it and each element has a pressure load, then the external force contribution at the node would be summed from the pressure load for all four elements.

Once this variable for the external force contribution from a pressure load is specified, it may be used like any other nodal variable. The user can, for example, specify the variable as a nodal variable to be output in a `RESULTS OUTPUT` command block. Or the user can reference the variable in a user subroutine.

7.6.1.6 Additional Commands

These command lines in the `PRESSURE` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the function command or the user subroutine. For example, if the magnitude of the pressure in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the pressure from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

Generally if multiple pressure sources are defined they are multiplied together. For example if all of the `FUNCTION`, `SCALE FACTOR`, and `READ VARIABLE` commands were defined the total pressure at each face would be the value of the function at the current time, times the value of the mesh variable at the specific faces, times the scaling factor. However, Some combinations of sources conflict and are not allowed to be used in the same boundary condition such as `SUBROUTINE` and `READ VARIABLE`.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.6.1.7 Surface Integration Schemes

```
INTEGRATION METHOD =
CODE_SELECTED|MULTILINEAR|QUADRATIC(CODE_SELECTED)
```

By default Sierra/SM will pick the pressure integration method that is most consistent with the underlying element type. For example using a quadratic pressure integration scheme on the face of a twenty seven node hexahedron and a linear pressure integration scheme on the face of eight node hexahedron. The higher order tetrahedra elements present some special cases. Fully integrated ten node tetrahedra are more consistent with a quadratic surface integration scheme and by default will use a quadratic pressure integration. The composite or mean quadrature ten node tetrahedra are more consistent with a multi-linear surface integration scheme and by default will use the linear integration method. An optional command `INTEGRATION METHOD` can be used to override this default behavior and explicitly set the integration scheme for pressure on six node triangle faces.

7.6.1.8 Example

```
BEGIN FUNCTION One
  TYPE = CONSTANT
  BEGIN VALUES
    1.0
  END VALUES
```

```
END FUNCTION One

BEGIN PRESSURE
  SURFACE = sideset_1
  FUNCTION = One
  SCALE FACTOR =1
END PRESSURE
```

This `PRESSURE` applies a force vector with a magnitude of 1.0 times the area associated with each node in `sideset_1` as shown in Figure 7.1. The force vectors are normal to the surface at the point which they are applied.

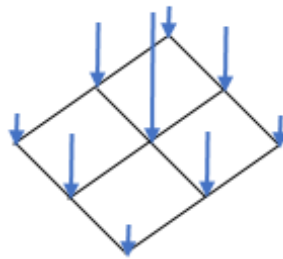


Figure 7.1: Result of a unit pressure load

7.6.2 Traction

```
BEGIN TRACTION
#
# surface set commands
SURFACE = <string list>surface_names
REMOVE SURFACE = <string list>surface_names
BLOCK = <string list> blocks
INCLUDE ALL BLOCKS
#
# specification commands
DIRECTION = <string>direction_name
FUNCTION = <string>function_name
#
# external traction sources
READ VARIABLE = <string>var_name
TIME = <real>time|FIRST|LAST
OBJECT TYPE = <string>FACE|NODE(FACE)
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SURFACE SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
#
# surface integration schemes
TRACTION INTEGRATION METHOD =
    NODAL_LUMPING|GAUSS_QUADRATURE
END [TRACTION]
```

The **TRACTION** command block applies a traction to each face in the associated surfaces. The traction has units of force per unit area. (A traction, unlike a pressure, may not necessarily be in the direction of the normal to the face to which it is applied.) The given traction is integrated over the surface area of a face.

The traction field can be determined by a **SIERRA** function or a user subroutine. If the traction field is constant over the faces and has only a time-varying component, the specification commands in the above command block can be used to specify the traction field. If the traction field has both

time-varying and spatially varying components, a user subroutine is used to specify the traction field.

The traction force is integrated in the current deformed model configuration. The value of the traction force is the value of the traction field defined by this boundary condition times the face area. Thus if due to displacement a face grows or shrinks the net force produced by the traction boundary condition will change. As the traction field is defined explicitly with a direction a rotation of a face will not change the traction force on that face.

The traction field can only be controlled by one method. Accordingly, a `TRACTION` command block can only contain one of the options: function or user subroutine.

Currently, the `TRACTION` command block can be used for surfaces that have faces derived from solid elements (eight-node hexahedra, four-node tetrahedra, eight-node tetrahedra, etc.), membranes, and shells.

The `TRACTION` command block contains three groups of commands surface set, specification, and user subroutine. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with the specification commands or the user subroutine option. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this force boundary condition for certain time periods. Following are descriptions of the different command groups.

7.6.2.1 Surface Set Commands

The `surface set commands` portion of the `TRACTION` command block defines a set of surfaces associated with the traction field. The surface specification commands work the same as for the pressure boundary condition as described in section [7.6.1.1](#).

7.6.2.2 Traction Specification Commands

If the direction command is used, the traction vector at any given time is the same for all surfaces associated with the particular `TRACTION` command block. The direction of the traction vector is constant for all time; the magnitude of the traction vector may vary with time, however.

Following are the command lines used to specify the traction with a direction and a function:

```
DIRECTION = <string>defined_direction
FUNCTION = <string>function_name
```

The traction is specified in an arbitrary user-defined direction, and is defined using the `DIRECTION` command line. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the `SIERRA` scope. Note that the direction is always a unit vector, even if it is specified with a non-unit vector in the `DEFINE DIRECTION` command block.

The magnitude of the traction is specified by the `FUNCTION` command line. This references a `function_name` (defined in the `SIERRA` scope in a `FUNCTION` command block) that specifies

the magnitude of the traction vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in Section 7.6.3.1.

7.6.2.3 User Subroutine Commands

If the user subroutine option is used, the traction vector may vary spatially at any given time for each of the surfaces associated with the particular `TRACTION` command block. The user subroutine option allows for a more complex description of the traction field than does the function option, but also requires writing a user subroutine to implement this capability.

A node set user subroutine will be used to define a traction at each node attached to a face which the boundary condition will be applied. This nodal traction is then integrated over the faces. A surface subroutine can be used to define a constant traction on each face. The subroutines will be called by Sierra/SM at the appropriate time to generate the traction field.

Following is the command line related to the user subroutine option:

```

NODE SET SUBROUTINE = <string>subroutine_name
SURFACE SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

The user subroutine option is invoked by using the `NODE SET` or `SURFACE` subroutine command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user. Associating traction values with nodes requires the use of a `NODE SET SUBROUTINE` command line. Associating traction values with faces requires the use of a `NODE SET SUBROUTINE` command line.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.6.3.1.

7.6.2.3.1 Usage requirements for the node set subroutine. The provided subroutine will return three values per node or face. Suppose the user subroutine provided passes the output values through an array `output_values`. For a given node `inode`, the `output_values` array would have the following values:

```
output_values(1,inode) = x component of traction vector
```

```
output_values(2,inode) = y component of traction vector
output_values(2,inode) = z component of traction vector
```

The values of the flags array are not used in the traction subroutine.

See Chapter 11 for more details on implementing the user subroutine option.

7.6.3 External Mesh Database Commands

The traction distribution may be defined by a field on the mesh database. When using a mesh database defined distribution the traction magnitude and/or direction is read from a finite element model defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model is the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3).

```
READ VARIABLE = <string>var_name
TIME = <real>time|FIRST|LAST
OBJECT TYPE = <string>FACE|NODE(FACE)
```

See Section 7.1.3 for more details.

The `OBJECT TYPE` command line specifies whether the traction field on the mesh file is specified for nodes (the mesh object type is `NODE`) or for faces (the mesh object type is `FACE`). If the `OBJECT TYPE` command line is not present, it is assumed that the named variable is defined on faces.

The `READ VARIABLE` command gives the mesh file variable which defines the traction. There are two ways to define this traction field: the first, is to define a field with a single scalar value for each node or face and couple that with a `DIRECTION` command to define the traction direction; the second, is to define a vector field with three components. In the latter case, the field can define a unique traction magnitude and direction at each node or face. Furthermore, this option should not be used with the `DIRECTION` command.

If present, the `TIME` command defines the time slice in the input mesh file from which to read the traction field. If the `TIME` field is not present, the traction will be defined by interpolation between time slices closest to the current analysis time. Specifying the keyword `FIRST` or `LAST` will read from the first or last time slice in the input mesh file, respectively.

7.6.3.1 Additional Commands

These command lines in the `TRACTION` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification

commands or the user subroutine. For example, if the magnitude of the traction in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the traction from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.6.3.2 Surface Integration Schemes

```
TRACTION INTEGRATION METHOD =
    NODAL_LUMPING | GAUSS_QUADRATURE
```

By default, Sierra/SM will automatically choose the traction integration method. This input command can over-ride the default behavior for most face topologies. The exceptions are eight- and nine-noded quadrilateral faces (e.g., the faces of higher-order hexahedron elements), which always integrate tractions with full-order Gauss quadrature. For all other face topologies, the default traction calculation method is `NODAL_LUMPING`: the traction evaluated at the nodal coordinates is multiplied by an area attributed to that node to get the nodal force contribution. This default can be replaced by full-order Gauss quadrature for any face topology by specifying `GAUSS_QUADRATURE` as the traction integration method.

7.6.3.3 Example

```
DEFINE DIRECTION Fdir WITH VECTOR -1 0 -1

BEGIN FUNCTION One
    TYPE = CONSTANT
    BEGIN VALUES
        1.0
    END VALUES
END FUNCTION One

BEGIN TRACTION
    SURFACE = sideset_1
    DIRECTION = Fdir
    FUNCTION = One
    SCALE FACTOR =1
END TRACTION
```

This `TRACTION` applies a force vector with a magnitude of 1.0 time by the area associated with each node in `sideset_1` as shown in Figure 7.2. The force vectors are not required to be normal to the surface.

7.6.4 Prescribed Force

```
BEGIN PRESCRIBED FORCE
#
# node set commands
```

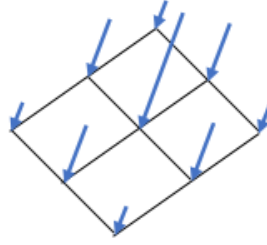


Figure 7.2: Result of a unit traction load

```

NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
DIRECTION = <string>defined_direction |
    COMPONENT = <string>X|Y|Z
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED FORCE]

```

The `PRESCRIBED FORCE` command block prescribes a force field for a given set of nodes. The force field associates a vector giving the magnitude and direction of the force with each node in the node set. The force field may vary over time and space. If the force field has only a time-varying component, the specification commands in the above command block can be used to specify the force field. If the force field has both time-varying and spatially varying components, a user subroutine is used to specify the force field. In a given boundary condition command block,

commands from only one of the command groups (specification commands or user subroutine commands) may be used.

The `PRESCRIBED FORCE` command block contains three groups of commands—node set, function, and user subroutine. Each of these command groups is basically independent of the others. In addition to the command lines in the three command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with either the specification commands or the user subroutine commands. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this force boundary condition for certain time periods. Following are descriptions of the different command groups.

7.6.4.1 Node Set Commands

The node set commands portion of the `PRESCRIBED FORCE` command block defines a set of nodes associated with the prescribed force field and can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.6.4.2 Specification Commands

If the specification commands are used, the force vector at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED FORCE` command block. The direction of the force vector is constant for all time; the magnitude of the force vector may vary with time, however.

Following are the command lines used to specify the prescribe force with a direction and a function:

```
DIRECTION = <string>defined_direction |
COMPONENT = <string>X|Y|Z
FUNCTION = <string>function_name
```

The force can be specified either along an arbitrary user-defined direction or along a component direction (X, Y, or Z), but not both.

- The `DIRECTION` command line is used to prescribe force in an arbitrary user-defined direction. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the SIERRA scope.
- The `COMPONENT` command line is used to specify that the force vector lies along one of the component directions. The `COMPONENT` command line is a shortcut to an internally defined direction vector; for example, component `x` corresponds to using direction vector (1, 0, 0).

The magnitude of the force is specified by the `FUNCTION` command line. This references a `function_name` (defined in the SIERRA scope in a `FUNCTION` command block) that specifies the magnitude of the force vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in Section 7.6.4.4.

The force is applied only in the prescribed direction, and is not applied in any direction orthogonal to that direction.

7.6.4.3 User Subroutine Commands

If the user subroutine option is used, the force vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED FORCE` command block. The user subroutine option allows for a more complex description of the force field than does the function option, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a force direction and a magnitude for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the force field.

Following are the command lines related to the user subroutine option:

```

NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value

```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.6.4.4.

7.6.4.3.1 Usage requirements for the node set subroutine. The provided subroutine will return three output values per node. Suppose a user subroutine is provided that passes the output values through an array `output_values`. For a given node `inode`, the `output_values` array would have the following values:

```
output_values(1,inode) = x component of force at inode
output_values(2,inode) = y component of force at inode
output_values(3,inode) = z component of force at inode
```

The three components of the force vector are given in `output_values` 1 through 3. The values of the flags array are ignored.

See Chapter 11 for more details on implementing the user subroutine option.

7.6.4.4 Additional Commands

These command lines in the `PRESCRIBED FORCE` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification commands or the user subroutine. For example, if the magnitude of the force in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the force from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.6.4.5 Example

```
DEFINE DIRECTION Fdir WITH VECTOR -1 0 -1

BEGIN FUNCTION One
  TYPE = CONSTANT
  BEGIN VALUES
    1.0
  END VALUES
END FUNCTION One

BEGIN PRESCRIBED FORCE
  SURFACE = sideset_1
  DIRECTION = Fdir
```

```

FUNCTION = One
SCALE FACTOR =1
END PRESCRIBED FORCE

```

The resulting PRESCRIBED FORCE applies a force vector of magnitude 1.0 to each node in sideset_1 as shown in Figure 7.3.

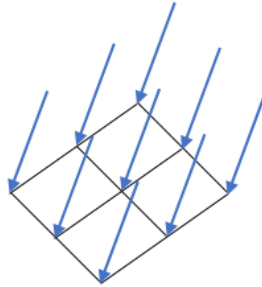


Figure 7.3: Result of a unit prescribed force

7.6.5 Prescribed Moment

```

BEGIN PRESCRIBED MOMENT
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
RIGID BODY = <string list>rigid_body_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# specification commands
DIRECTION = <string>defined_direction |
    COMPONENT = <string>X|Y|Z
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#

```

```

# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED MOMENT]

```

The `PRESCRIBED MOMENT` command block prescribes a moment field for a given set of nodes. Moments can only be defined for nodes with rotational degrees of freedom. This includes nodes attached to beam and shell elements, rigid body nodes and potentially super element nodes.

The moment field associates a vector giving the magnitude and direction of the moment with each node in the set. If the moment field has only a time-varying component, the specification commands in the above command block can be used to specify the moment field. If the moment field has both time-varying and spatially varying components, a user subroutine option is used to specify the moment field. In a given boundary condition command block, commands from only one of the command groups (specification commands or user subroutine commands) may be used.

The `PRESCRIBED MOMENT` command block contains three groups of commands—node set, function, and user subroutine. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with either the specification commands or the user subroutine commands. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this force boundary condition for certain time periods. Following are descriptions of the different command groups.

7.6.5.1 Node Set Commands

The `node set commands` portion of the `PRESCRIBED MOMENT` command block defines a set of nodes associated with the prescribed moment field and can include some combination of the following command lines:

```

NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.6.5.2 Specification Commands

If the specification commands are used, the moment vector at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED MOMENT` command block. The direction of the moment vector is constant for all time; the magnitude of the moment vector may vary with time, however.

Following are the command lines used to specify the prescribed moment with a function and a direction:

```
DIRECTION = <string>defined_direction |  
  COMPONENT = <string>X|Y|Z  
FUNCTION = <string>function_name
```

The moment can be specified either along an arbitrary user-defined direction or along a component direction (X, Y, or Z), but not both.

- The `DIRECTION` command line is used to prescribe the moment in an arbitrary user-defined direction. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the SIERRA scope.
- The `COMPONENT` command line is used to specify that the moment vector lies along one of the component directions. The `COMPONENT` command line is a shortcut to an internally defined direction vector; for example, component `x` corresponds to using direction vector (1, 0, 0).

The magnitude of the moment is specified by the `FUNCTION` command line. This references a `function_name` (defined in the SIERRA scope in a `FUNCTION` command block) that specifies the magnitude of the moment vector as a function of time. The magnitude can be scaled by use of the `SCALE FACTOR` command line described in Section 7.6.5.4.

The moment is applied only in the prescribed direction, and is not applied in any direction orthogonal to that direction.

7.6.5.3 User Subroutine Commands

If the user subroutine option is used, the moment vector may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED MOMENT` command block. The user subroutine option allows for a more complex description of the moment field than do specification commands, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a moment direction and a magnitude for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the moment field.

Following are the command lines related to the user subroutine option:

```
NODE SET SUBROUTINE = <string>subroutine_name  
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON  
SUBROUTINE REAL PARAMETER: <string>param_name
```

```

    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value

```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the `NODE SET SUBROUTINE` command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided throughout Chapter 11.

The magnitude set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.6.5.4.

7.6.5.3.1 Usage requirements for the node set subroutine. The provided subroutine will return three output values per node. Suppose a provided user subroutine passes the output values through an array `output_values`. For a given node `inode`, the `output_values` array would have the following values:

```

output_values(1,inode) = moment about x-direction at inode
output_values(2,inode) = moment about y-direction at inode
output_values(3,inode) = moment about z-direction at inode

```

The three components of the moment vector are given in `output_values` 1 through 3. The values of the flags array are ignored.

See Chapter 11 for more details on implementing the user subroutine option.

7.6.5.4 Additional Commands

These command lines in the `PRESCRIBED MOMENT` command block provide additional options for the boundary condition:

```

SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names

```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all vector magnitude values of the field defined by the specification commands or the user subroutine. For example, if the magnitude of the moment in a time history function is given as 1.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the magnitude of the moment from time 1.0 to 2.0 is 0.75. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.7 Distributed Force and Moment

```
BEGIN DISTRIBUTED FORCE|MOMENT
#
# Node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
ELEMENT = <int list>elem_numbers
NODE = <int list>node_numbers
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# Load definition commands
SCALE FACTOR = <real>scale_factor(1.0)
FUNCTION = <string>function_name
DIRECTION = <string>defined_direction |
    !COMPONENT = <string>X|Y|Z
COORDINATE SYSTEM = <string>sys_name(global)
#
# additional commands
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [DISTRIBUTED FORCE|MOMENT]
```

The distributed force and moment boundary conditions are used to apply a specific total net force or moment that is automatically distributed to a finite set of nodes. The distributed force and moment boundary conditions differ from the prescribed force [7.6.4](#) and moment [7.6.5](#) boundary conditions. The previously described prescribed force boundary conditions apply user defined point loads to each and every node used by the boundary condition. The distributed force BC instead spreads a single user defined target load among every node in the set. The distributed moments are applied via translational force couples and thus can be used on any element type. This differs from the prescribed point load moments [7.6.5](#) which can only be used on elements with rotational degrees of freedom such as shells.

The distributed load boundary conditions are targeted to the following types of use cases.

- Apply a moment to a solid meshed body, for example torque on a shaft or bending moment on the end of a solid meshed bar.
- Apply an exact known force to a model boundary. For example to model a load-control tension test.
- Apply a load to known load to a model distributed over a finite volume. An analysis with this distributed load will converge with mesh refinement. An analysis with a concentrated load applied to a single point will generate a singularity at that point and not converge with mesh refinement.

7.7.0.5 Node Set Commands

The Node set commands portion of the DISTRIBUTED FORCE and DISTRIBUTED MOMENT command blocks define a set of nodes associated with the boundary condition. These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines. The boundary condition must contain at least one NODE SET, NODE, SURFACE, BLOCK, ELEMENT, or INCLUDE ALL BLOCKS command line. Note, the RIGID BODY command is not available with this boundary condition.

7.7.0.6 Load Definition Commands

The load commands define the magnitude and direction of the force or moment applied by the distributed load boundary condition. The load magnitude is defined by the time history function `function_name` multiplied by the value of `scale_factor`. Each distributed load BC takes a single load magnitude, the FUNCTION command by default takes a function of time and may also take a analytic function that involves other global variables. One and only one of the COMPONENT command line or DIRECTION command line must be given to define the load direction. For the distributed force BC the load will be in the given direction. For the distributed moment BC the load will be around the given direction using the right hand rule. If no coordinate system is given the direction or component refer to the global XYZ coordinate system. If an optional coordinate system is specified the directions and components are given with respect to that coordinate system. Note if the coordinate system is dynamic (linked to coordinates of node sets) the direction of distributed load application will update continuously as the model evolves.

7.7.0.7 Miscellaneous Commands

The ACTIVE PERIODS and INACTIVE PERIODS command lines provides an additional option for the distributed load BC. These command lines can activate or deactivate the load for certain time periods. See Section 2.6 for more information about these command lines.

7.7.0.8 Restrictions and Assumptions

The distributed load boundary conditions are applied by defining a set of translational forces on the nodes in the application region. Generally forces will be mass proportional. Example force distributions for Z force and Z moment are shown in Figure 7.4 and 7.5. Force distributions are constructed to apply net load in only a single component. Thus the force distribution shown in Figure 7.4 produces no net moment on the surface and no net Force in the X and Y directions. The moment distribution shown in Figure 7.5 produces no net force on the surface nor any net moment around the X or Y axes.

As moments are constructed via translational force couples no net moment can be applied to a single node. Additionally if all the applied nodes fall nearly in a line no net moment can be applied around that line. If these degenerate cases occur the inapplicable component of the net moment will be zero. For node sets that are close to collinear potentially unphysical large forces may be required to apply due to the small axial moment arm. When using the distributed moment BC generally the application region should be a fairly large set of nodes on a block or surface to

prevent these excessive force concentrations.

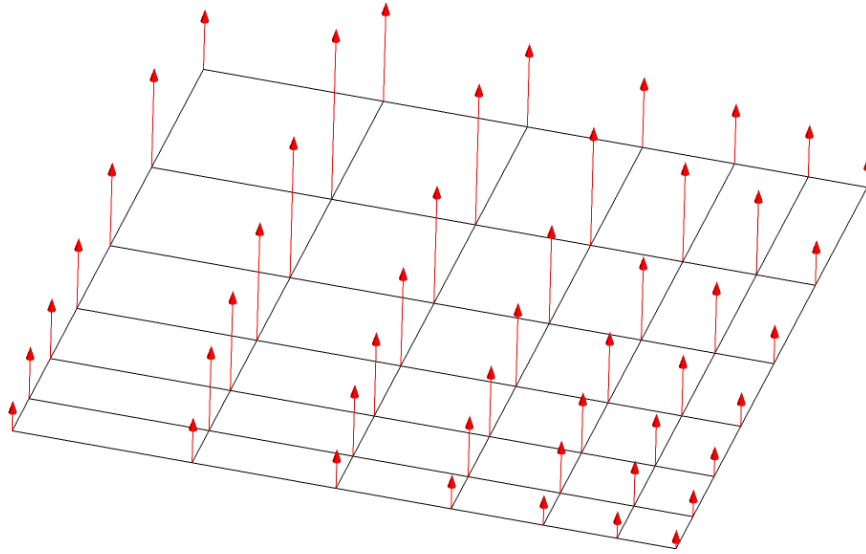


Figure 7.4: Distributed Z Force Load Distribution

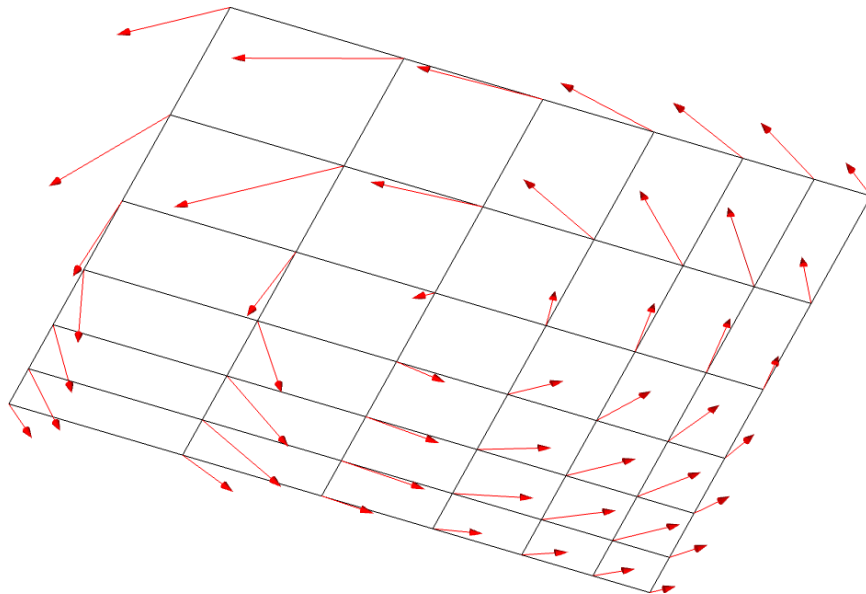


Figure 7.5: Distributed Z Moment Load Distribution

7.7.0.9 Example: Applying Torque to Solid Bar

The following example for applying torque around an arbitrarily oriented shaft shown in Figure 7.6. The coordinate system `rod_1` is linked to nodes on the body and will update as the rod moves so that the torque is always applied around the rod axis.

```
# Define a body fitted coordinate system on the rod
```

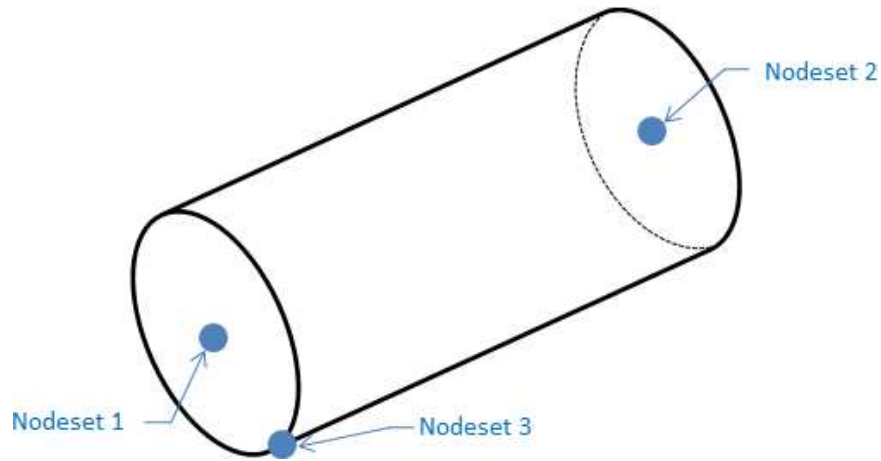



Figure 7.6: Distributed Load Example

```
begin coordinate system rod_1
  type    = rectangular
  # Base of Cylinder
  origin node = nodelist_1
  # Vector from origin to vector nodes defines Z axis
  vector node = nodelist_2
  # A point on the XY plane
  point  node = nodelist_3
end

# Ramp up loading function
begin function ramp
  type is piecewise linear
  begin values
    0.00 0.0
    0.01 1.0
  end values
end

# Apply a distributed load to the rod block, around the
# axis of the cylindrical rod.
begin distributed moment
  block = block_1
  coordinate system = rod_1
  function = ramp
  # Z coordinate direction of coordinate system lies along
  # the rod axis
  component = z
end
```

7.8 Gravity

```
BEGIN GRAVITY
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
DIRECTION = <string>defined_direction
COMPONENT = <string>X|Y|Z

FUNCTION = <string>function_name
GRAVITATIONAL CONSTANT = <real>g_constant
SCALE FACTOR = <real>scale_factor(1.0)
EXTERNAL FORCE CONTRIBUTION OUTPUT NAME =
    <string>variable_name
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [GRAVITY]
```

A gravity load is generally called a body force boundary condition. A gravity load generates a force at a node that is proportional to the mass of the node. This section describes how to apply a gravity load to a body.

The `GRAVITY` command block is used to specify a gravity load that is applied to all nodes selected within a command block. The gravity load boundary condition uses the function and scale (gravitational constant and scale factor) information to generate a body force at a node based on the mass of the node. Multiple `GRAVITY` command blocks can be defined on different sets of nodes. If two different `GRAVITY` command blocks reference the same node, the node will have gravity loads applied by both of the command blocks. Care must be taken to make sure multiple gravity loads are not applied to one block if only one gravity load condition is desired.

The `node set commands` portion of the `GRAVITY` command block defines a set of nodes associated with the gravity load and can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` command line in the command block.

The gravity load is specified along an arbitrary user-defined direction, and is defined using the `DIRECTION` command line. The name in the string `defined_direction` is a reference to a direction, which is defined using the `DEFINE DIRECTION` command block within the `SIERRA` scope. Alternatively the gravity can be applied along the `X`, `Y`, or `Z` `COMPONENT` directions.

The strength of the gravitational field can be varied with time by using the `FUNCTION` command line. This command line references a `function_name` defined in the `SIERRA` scope in a `FUNCTION` command block. The gravity force is constant in time if the `FUNCTION` command line is not specified.

A gravitational constant is specified by the `GRAVITATIONAL CONSTANT` command line in the real value `g_constant`. For example, the gravitational constant in units of inches and seconds would be 386.4 inches per second squared. This quantity must be set based on the actual units for the model.

The dependent variables in the function can be scaled by the real value `scale_factor` in the `SCALE FACTOR` command line. At any given time, the strength of the gravitational field is a product of the gravitational constant, the value of the function at that time, and the scale factor.

If the `EXTERNAL FORCE CONTRIBUTION OUTPUT NAME` command line appears there will be a variable created with whatever name the user specifies for `variable_name`. The variable defines a three-dimensional vector at each node associated with this particular command block. The three-dimensional vector at each node represents the external force due solely to just this particular external force boundary contribution.

Once this variable for the external force contribution is specified, it may be used like any other nodal variable. The user can, for example, specify the variable as a nodal variable to be output in a `RESULTS OUTPUT` command block. Or the user can reference the variable in a user subroutine.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines provides an additional option for the gravity load condition. These command lines can activate or deactivate the gravity load for certain time periods. See Section 2.6 for more information about these command lines.

7.9 Centripetal Force

```
BEGIN CENTRIPETAL FORCE
  NODE SET = <string list>nodelist_names
  SURFACE = <string list>surface_names
  BLOCK = <string list>block_names
  INCLUDE ALL BLOCKS
  REMOVE NODE SET = <string list>nodelist_names
  REMOVE SURFACE = <string list>surface_names
  REMOVE BLOCK = <string list>block_names
  #
  CYLINDRICAL AXIS = <string>cylindrical_axis
  ROTATIONAL VELOCITY FUNCTION = <string>rotational_vel_name
  ROTATIONAL VELOCITY SCALE FACTOR = <real>rvsf(1.0)
  FORCE SCALE FACTOR = <real>fsf(1.0)
  ACTIVE PERIODS = <string list>period_names
  INACTIVE PERIODS = <string list>period_names
END [CENTRIPETAL FORCE]
```

The centripetal force boundary condition adds an external force to the body that is equivalent to the centripetal force on the body rotating at the defined rotational velocity. The centripetal force boundary condition can be used, for example, in a static analysis to calculate the deformation of a body spinning with constant rotational velocity around some axis.

The centripetal force boundary condition will be applied to all nodes selected within the command block. The equation for the centripetal force on a node is given by Equation 7.4, where F is the magnitude of the centripetal force, m is the mass of the node, R is the current distance from the node to the rotational axis, and ω is the rotational velocity of the node about the rotational axis.

$$F = mR\omega^2 \quad (7.4)$$

The `CYLINDRICAL AXIS` command defines the name of the axis of rotation. The name given is the name of a previously defined axis created with the `DEFINE AXIS` command. The rotational velocity as a function of time is defined by multiplying the function named by the `ROTATIONAL VELOCITY FUNCTION` by the `ROTATIONAL VELOCITY SCALE FACTOR`. Additionally a scale factor on the final return force may be specified with the `FORCE SCALE FACTOR` command.

7.10 Prescribed Temperature

```
BEGIN PRESCRIBED TEMPERATURE
#
# block set commands
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
ELEMENT = <int_list>elem_ids

NODE SET = <string list>nodeset_names
REMOVE NODE SET = <string list>nodeset_names
NODE = <int_list>node_ids
#
# specification command
FUNCTION = <string>function_name
#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
# external database commands
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
TEMPERATURE TYPE = SOLID_ELEMENT|SHELL_ELEMENT (SOLID_ELEMENT)
#
# coupled analysis commands
RECEIVE FROM TRANSFER [FIELD TYPE = NODE|ELEMENT (NODE)]
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PRESCRIBED TEMPERATURE]
```

The `PRESCRIBED TEMPERATURE` command block prescribes a temperature field for a given set of nodes. The prescribed temperature is for each node in the node set. The temperature field may vary over time and space. If the temperature field has only a time-varying component, the function command in the above command block can be used to specify the temperature field. If the temperature field has both time-varying and spatially varying components, a user subroutine option can be used to specify the temperature field. Finally, the temperature may also be read as a variable

from the mesh file. Only one of these options—function, user subroutine, or read variable—may be specified in a command block.

Temperature is applied to nodes, but it is frequently used at the element level, such as in the case for thermal strains. If the temperatures are used at the element level, the nodal values are averaged (depending on element) connectivity to produce an element temperature. The temperatures must be defined for all the nodes defining the connectivity for any given element. For this reason, we use block commands to derive a set of nodes at which to define temperatures. If the temperatures are used on an element basis, then the temperature at all the necessary nodes will be defined.

The `PRESCRIBED TEMPERATURE` command block contains four groups of commands—block set, function, user subroutine, and read variable. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The `SCALE FACTOR` command line can be used in conjunction with the function command, the user subroutine option, or the read variable option. The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate this kinematic boundary condition for certain time periods. Following are descriptions of the different command groups.

7.10.1 Block Set Commands

The `block set commands` portion of the `PRESCRIBED TEMPERATURE` command block defines a set of nodes associated with the prescribed temperature field and can include some combination of the following command lines:

```
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
ELEMENT = <int_list>elem_ids
NODE SET = <string list>nodeset_names
REMOVE NODE SET = <string list>nodeset_names
NODE = <int_list>node_ids
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes derived from some combination of element blocks. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `BLOCK`, `INCLUDE ALL BLOCKS`, `ELEMENT`, `NODE SET`, or `NODE` command line in the command block.

7.10.2 Specification Command

If the function command is used, the temperature at any given time is the same for all nodes in the node set associated with the particular `PRESCRIBED TEMPERATURE` command block. The command line

```
FUNCTION = <string>function_name
```

references a `function_name` (defined in the SIERRA scope using a `FUNCTION` command block) that specifies the temperature as a function of time. The temperature can be scaled by use of the `SCALE FACTOR` command line described in Section 7.10.6.

7.10.3 User Subroutine Commands

If the user subroutine option is used, the temperature field may vary spatially at any given time for each of the nodes in the node set associated with the particular `PRESCRIBED TEMPERATURE` command block. The user subroutine option allows for a more complex description of the temperature field than does the function command, but also requires writing a user subroutine to implement this capability. The user subroutine will be used to define a temperature for every node to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the temperature field.

Following are the command lines related to the user subroutine option:

```

NODE SET SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Several other commands control the behavior of user subroutines: `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. These are described in Section 11.2.2. Examples of using these command lines are provided throughout Chapter 11.

The temperature set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.10.6.

See Chapter 11 for more details on implementing the user subroutine option.

7.10.4 External Mesh Database Commands

```

READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
TEMPERATURE TYPE = SOLID_ELEMENT|SHELL_ELEMENT (SOLID_ELEMENT)
```

See Section 7.1.3 for more details.

If temperature is to be prescribed for shell elements, a linear or quadratic thermal gradient can optionally be specified through the thickness of the shells using the `TEMPERATURE TYPE` command line. The `SOLID_ELEMENT` option to this command is the default behavior, and results in a single temperature being read for each node into the `temperature` variable. With the `SHELL_ELEMENT` option, temperatures are read into the `shell_temperature` variable. Shell elements may potentially define a temperature gradient through the thickness, in which case there will be multiple temperatures at a node to describe the temperature gradient through the shell. The `TEMPERATURE_TYPE` command is only valid in conjunction with the `READ VARIABLE` command.

Though-thickness shell temperatures follow the Aria convention. If there is a single shell temperature defined at a node, the temperature is constant through the thickness.

If there are two shell temperatures defined at a node, the first is the temperature on the bottom of the shell and the second the temperature at the top. The temperature varies linearly between the top and bottom.

If there are three shell temperatures defined at node, the first is the temperature at the bottom of the shell, the second the temperature at the middle of the shell, and the third the temperature at the top of the shell. The temperature varies quadratically through the thickness.

`SOLID_ELEMENT` and `SHELL_ELEMENT` temperatures may be defined simultaneously in the same analysis through two different temperature command blocks. If both are defined, the shell element temperature results override any solid element temperature results on the shell elements.

7.10.5 Coupled Analysis Commands

The `RECEIVE FROM TRANSFER` command provides the ability to set the temperature in a coupled analysis by transferring results from another SIERRA code, such as Aria.

```
RECEIVE FROM TRANSFER [FIELD TYPE = NODE|ELEMENT(NODE)]
```

If this command is used in its default form, it is expected that the temperature is transferred to a nodal field named `temperature` in the Sierra/SM region. Sierra/SM performs an interpolation from the nodal field to an element field named `temperature`. The temperature can also be transferred directly to the element `temperature` field by using the optional `FIELD TYPE = ELEMENT` argument to this command.

If the `RECEIVE FROM TRANSFER` command is used, but the appropriate commands to perform the transfer between the two regions are missing, the temperature will be zero during the entire simulation.

It is also possible to use this line command to cause the initial temperatures obtained from a restart file to remain constant in time.

7.10.6 Additional Commands

These command lines in the `PRESCRIBED TEMPERATURE` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
```



```
ACTIVE PERIODS = <string list>period_names  
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all temperature values of the field defined by the function command, the user subroutine, or the read variable option. For example, if the temperature in a time history function is given as 100.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the temperature from time 1.0 to 2.0 is 50.25. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See [Section 2.6](#) for more information about these command lines.

7.11 Pore Pressure

```
BEGIN PORE PRESSURE
#
# block set commands
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
#
# specification command
FUNCTION = <string>function_name
#
# user subroutine commands
ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
# external database commands
READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST
#
# coupled analysis commands
RECEIVE FROM TRANSFER [FIELD TYPE = NODE|ELEMENT (NODE)]
#
# additional commands
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [PORE PRESSURE]
```

The `PORE PRESSURE` command block prescribes a pore pressure field for a given set of elements. The pore pressure is prescribed for each element in the block. The pore pressure field may vary over time and space. If the pore pressure field has only a time-varying component, the function command in the above command block can be used to specify the pore pressure field. If the pore pressure field has both time-varying and spatially varying components, a user subroutine option can be used to specify the pore pressure field. Finally, the pore pressure can be read as a variable from the mesh file. Only one of these options—function, user subroutine, or read variable—may be specified in a command block.

The `PORE PRESSURE` command block contains four groups of commands—block set, function, user subroutine, and read variable. Each of these command groups is basically independent of the others. In addition to the command lines in the four command groups, there are three additional command lines: `SCALE FACTOR`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. The

SCALE FACTOR command line can be used in conjunction with the function command, the user subroutine option, or the read variable option. The ACTIVE PERIODS and INACTIVE PERIODS command lines are used to activate or deactivate this kinematic boundary condition for certain time periods. Following are descriptions of the different command groups.

Biot's coefficient can be defined when prescribing pore pressure. See Section 5.1.2 for more information on Biot's coefficient.

7.11.1 Block Set Commands

The block set commands portion of the PORE PRESSURE command block defines a set of elements associated with the pore pressure field and can include a combination of the following command lines:

```
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of elements derived from some combination of element blocks. See Section 7.1.1 for more information about the use of these command lines for creating a set of elements used by the boundary condition. There must be at least one BLOCK or INCLUDE ALL BLOCKS command line in the command block.

7.11.2 Specification Command

If the FUNCTION command is used, the pore pressure at any given time is the same for all elements in the element set associated with the particular PORE PRESSURE command block. The command line

```
FUNCTION = <string>function_name
```

references a function_name (defined in the SIERRA scope using a FUNCTION command block) that specifies the pore pressure as a function of time. The pore pressure can be scaled using the SCALE FACTOR command line described in Section 7.11.6.

7.11.3 User Subroutine Commands

If the user subroutine option is used, the pore pressure field may vary spatially at any given time for each of the elements in the element set associated with the particular PORE PRESSURE command block. The user subroutine option allows for a more complex description of the pore pressure field than does the FUNCTION command, but the user subroutine option also requires that a user subroutine be written to implement this capability. The user subroutine will be used to define a pore pressure for every element to which the boundary condition will be applied. The subroutine will be called by Sierra/SM at the appropriate time to generate the pore pressure field.

Following are the command lines related to the user subroutine option:

```

ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value

```

The user subroutine option is invoked by using the `ELEMENT BLOCK SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Several other commands control the behavior of user subroutines: `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. These are described in Section 11.2.2. Examples of using these command lines are provided throughout Chapter 11.

The pore pressure set in the user subroutine can be scaled by use of the `SCALE FACTOR` command line, as described in Section 7.11.6.

See Chapter 11 for more details on implementing the user subroutine option.

7.11.4 External Mesh Database Commands

The pore pressure field can be read from an external mesh database. The finite element model from which pore pressures are read is defined via the `FINITE ELEMENT MODEL` command block described in Section 6.1. The finite element model can either be the model used by the region for its mesh definition as specified with the `USE FINITE ELEMENT MODEL` command (see Section 2.3), or it can be a different (but compatible) model. The following command lines control the use of an external mesh database to prescribe the pore pressure:

```

READ VARIABLE = <string>var_name
COPY VARIABLE = <string>var_name [FROM MODEL <string>model_name]
    MAP_BY_PROXIMITY|MAP_BY_ID (MAP_BY_PROXIMITY)
TIME = <real>time|FIRST|LAST

```

See Section 7.1.3 for more details.

7.11.5 Coupled Analysis Commands

The `RECEIVE FROM TRANSFER` command provides the ability to set the pore pressure in a coupled analysis by transferring results from another SIERRA code, such as Aria.

```

RECEIVE FROM TRANSFER [FIELD TYPE = NODE|ELEMENT (NODE)]

```

If this command is used in its default form, it is expected that the pore pressure is transferred to a nodal field named `pore_pressure` in the Sierra/SM region. Sierra/SM performs an interpolation

from the nodal field to an element field named `pore_pressure`. The pore pressure can also be transferred directly to the element `pore_pressure` field by using the optional `FIELD TYPE = ELEMENT` argument to this command.

If the `RECEIVE FROM TRANSFER` command is used, but the appropriate commands to perform the transfer between the two regions are missing, the pore pressure will be zero during the entire simulation.

7.11.6 Additional Commands

These command lines in the `PORE PRESSURE` command block provide additional options for the boundary condition:

```
SCALE FACTOR = <real>scale_factor(1.0)
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `SCALE FACTOR` command line is used to apply an additional scaling factor, which is constant in both time and space, to all pore pressure values of the field defined by the function command, the user subroutine, or the read variable option. For example, if the pore pressure in a time history function is given as 100.5 from time 1.0 to time 2.0 and the scale factor is 0.5, then the pore pressure from time 1.0 to 2.0 is 50.25. The default value for the scale factor is 1.0.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See [Section 2.6](#) for more information about these command lines.

7.12 Fluid Pressure

```
BEGIN FLUID PRESSURE
#
# surface set commands
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
#
# specification commands
DENSITY = <real>fluid_density
DENSITY FUNCTION = <string>density_function_name
GRAVITATIONAL CONSTANT = <real>gravitational_acceleration
FLUID SURFACE NORMAL = <string>direction_name
DEPTH = <real>fluid_depth
DEPTH FUNCTION = <string>depth_function_name
REFERENCE POINT = <string>reference_point_name
#
# additional commands
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
EXTERNAL FORCE CONTRIBUTION NAME = <string>varName
END [FLUID PRESSURE]
```

The `FLUID PRESSURE` command block applies a hydrostatic pressure to each node of each face in the associated surfaces. The pressure at any node is determined from

$$P = \rho gh \quad (7.5)$$

where P is the pressure, ρ is the fluid density at the current time, g is the gravitational constant, and h is the current depth of the fluid above the node. The depth of the fluid is computed as the distance from the node up to the current fluid surface in the direction of the fluid surface normal. The global location of the fluid surface is found by adding the current depth to the projection of a datum point along the direction defined in the `FLUID SURFACE NORMAL` command. The datum point is either the point specified in the `REFERENCE POINT` line command, or, in the absence of this command, the minimum point on the applied pressure surface in the direction defined in the `FLUID SURFACE NORMAL` command. Once the current location of the fluid surface is computed, the depth at each node on the pressure surface is computed as the distance from the node to the fluid surface in the direction of the fluid surface normal. See Figure 7.7.

Currently, the `FLUID PRESSURE` command block can be used for surfaces that have faces derived from solid elements (eight-node hexahedra, four-node tetrahedra, eight-node tetrahedra, etc.), membranes, and shells.

The `FLUID PRESSURE` command block contains three groups of commands—surface set, specification, and additional optional commands.

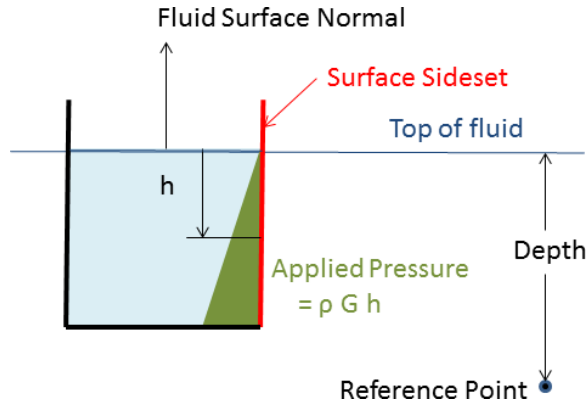


Figure 7.7: Fluid Pressure Setup

7.12.1 Surface Set Commands

The `surface set` commands portion of the `FLUID PRESSURE` command block defines a set of surfaces associated with the pressure field. The surface specification commands work the same as for the basic pressure boundary condition as described in section 7.6.1.1.

The force computed from the hydrostatic pressure will be in the opposite direction of the face normal. When using shells or membranes, the analyst must ensure that all face normals composing the pressure application surface are in the correct direction.

7.12.2 Specification Commands

The density and the gravitational acceleration must be input by the user in units consistent with other material properties and the lengths in the mesh. To facilitate convergence of the initial load step, the gradual application of a hydrostatic load may be specified through a time history function for the density. A combination of the `DENSITY` and the `DENSITY FUNCTION` sets the value for the fluid density at each time. Either the `DENSITY` or the `DENSITY FUNCTION` must be input and both can be used together. If the `DENSITY` command is input without the `DENSITY FUNCTION` command, the density will be constant in time at that value. If the `DENSITY FUNCTION` command is input without a `DENSITY` command, the function is used as a time history of the density. If both the `DENSITY` and the `DENSITY FUNCTION` commands are input, the density value is used as a scale factor on the time history function. Finally, the `GRAVITATIONAL CONSTANT` sets the value for the acceleration due to gravity, g .

```
DENSITY = <real>fluid_density
DENSITY FUNCTION = <string>fluid_density_function
GRAVITATIONAL CONSTANT = <real>G
```

The following command lines are used to define the location of the fluid surface at any time during the analysis:

```
FLUID SURFACE NORMAL = <string>normal_direction
```

```

DEPTH = <real>initial_fluid_depth
DEPTH FUNCTION = <string>depth_function_name
REFERENCE POINT = <string>point_name

```

The `FLUID SURFACE NORMAL` command sets the outward normal of the fluid surface. The fluid depth is then assumed to be in the direction opposite this global direction. The `DEPTH` command is used with the `DEPTH FUNCTION` command to determine the fluid depth at any time. At least one of these commands must be input. If the `DEPTH` command is input without the `DEPTH FUNCTION` command, the fluid depth will be constant in time with that value. If the `DEPTH FUNCTION` command is input without a `DEPTH` command, the function is used as a time history of the depth. If both the `DEPTH` and the `DEPTH FUNCTION` commands are input, the specified depth is used as a scale factor on the time history function.

The depth and/or depth function are used to determine the current depth, which is added to the appropriate position of a datum point to compute the current location of the fluid surface in the `FLUID SURFACE NORMAL` direction. Unless the optional `REFERENCE POINT` command described below is used, the datum point is chosen to be a point on a surface defined in the `SURFACE` command that minimizes the projection along the `FLUID SURFACE NORMAL` direction.

The `REFERENCE POINT` command line is used to specify the fluid surface relative to an external datum. When applying an external fluid pressure in a quasistatic analysis, a corresponding stiffness due to the external fluid is added to the diagonal terms of the stiffness matrix for the full tangent preconditioner to enhance convergence of the solver.

7.12.3 Additional Commands

These command lines in the `FLUID PRESSURE` command block provide additional options for the boundary condition:

```

EXTERNAL FORCE CONTRIBUTION OUTPUT NAME =
    <string>variable_name
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names

```

If the `EXTERNAL FORCE CONTRIBUTION OUTPUT NAME` command line appears there will be a variable created with whatever name the user specifies for `variable_name`. The variable defines a three-dimensional vector at each node associated with this particular command block. The three-dimensional vector at each node represents the external force due solely to just this particular external force boundary contribution.

Once this variable for the external force contribution is specified, it may be used like any other nodal variable. The user can, for example, specify the variable as a nodal variable to be output in a `RESULTS OUTPUT` command block. Or the user can reference the variable in a user subroutine.

By default, the `FLUID PRESSURE` boundary condition will be active throughout an analysis. However, use of the `ACTIVE PERIODS` and `INACTIVE PERIODS` commands can be used to limit the action of the boundary condition to specific time periods. The `ACTIVE PERIODS` command line specifies when the boundary condition is active implying that it is inactive during any periods

not included on the command line. Alternatively, the `INACTIVE PERIODS` determines when the boundary condition will not be active. See Section 2.6 for more information about these commands.

7.13 Inertia Relief

```
BEGIN INERTIA RELIEF
#
# node set commands
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
ELEMENT = <integer list>element_ids
NODE = <integer list>node_ids
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names
#
# component commands
COMPONENT = <string>X|Y|Z
# additional commands
EXTERNAL FORCE CONTRIBUTION OUTPUT NAME = <string>output_field
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [INERTIA RELIEF]
```

The `INERTIA RELIEF` command block computes a force or a moment in the given component directions that counteracts the external forces/moments in the problem. This capability is useful when a body is flying through space given an external force contribution, but the user would like the body to remain in a constant frame of reference.

The `INERTIA RELIEF` command block contains two groups of commands—node set and component. Each of these command groups is basically independent of the others. In addition to the command lines in the two command groups, there are three additional command lines: `EXTERNAL FORCE CONTRIBUTION OUTPUT NAME`, `ACTIVE PERIODS`, and `INACTIVE PERIODS`. Following are descriptions of the different command groups.

7.13.1 Node Set Commands

The `node set commands` portion of the `INERTIA RELIEF` command block defines a set of nodes associated with the inertia relief field and can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
```

```

ELEMENT = <integer list>element_ids
NODE = <integer list>node_ids
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list>surface_names
REMOVE BLOCK = <string list>block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes. See Section 7.1.1 for more information about the use of these command lines for creating a set of nodes used by the boundary condition. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, `ELEMENT`, `NODE`, or `INCLUDE ALL BLOCKS` command line in the command block.

7.13.2 Component Command

The `COMPONENT` command is used to specify the direction of the counterbalance force and is the same for all nodes in the node set associated with the particular `INERTIA RELIEF` command block. As an example, a component x corresponds to using direction vector (1, 0, 0). By default, all components are used in the inertia relief calculation.

7.13.3 Additional Commands

These command lines in the `INERTIA RELIEF` command block provide additional options for the boundary condition:

```

EXTERNAL FORCE CONTRIBUTION OUTPUT NAME =
    <string>name_of_output_field
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names

```

The `EXTERNAL FORCE CONTRIBUTION OUTPUT NAME` command line can be used to specify the name of the output variable to be computed that is the counterbalance force calculated inside the inertia relief code. Note, in order to output the variable, the variable must still be requested as a nodal variable in the necessary output sections.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the boundary condition is active. See Section 2.6 for more information about these command lines.

7.13.4 General Notes

- If using a single `INERTIA RELIEF` command block, all blocks in the model must be connected in some way. To apply inertia relief to separated blocks in the model, multiple `INERTIA RELIEF` command blocks must be defined for each separate block.
- Inertia relief does not counterbalance kinematic boundary conditions.
- It is not guaranteed that the displacement of the model is zero. The only guarantee is for the momentum to be zero or near zero.

7.13.5 Example

```
Begin Inertia Relief
  include all blocks
End Inertia Relief
```

The resulting `INERTIA RELIEF` applies forces and moments that balance out the external forces and moments of all blocks in the model. The net forces and moments applied by the inertia relief boundary condition are available for output in the global variables `force_inertia_relief` and `moment_inertia_relief`. The centroid about which moments are applied is placed in the global variable `centroid_inertia_relief`.

7.14 Specialized Boundary Conditions

Specialized boundary conditions that are provided to enforce kinematic conditions or apply loads are described in this section.



7.14.1 Cavity Expansion

```
BEGIN CAVITY EXPANSION
  SURFACE = <string list>surface_names
  BLOCK = <string list>block_list
  INCLUDE ALL BLOCKS
  REMOVE SURFACE = <string list>surface_names
  EXPANSION RADIUS = <string>SPHERICAL|CYLINDRICAL
    (SPHERICAL)
  FREE SURFACE = <real>top_surface_zcoord
    <real>bottom_surface_zcoord
  NODE SETS TO DEFINE BODY AXIS =
    <string>nodelist_1 <string>nodelist_id2
  TIP RADIUS = <real>tip_radius
  BEGIN LAYER <string>layer_name
    LAYER SURFACE = <real>top_layer_zcoord
      <real>bottom_layer_zcoord
    PRESSURE COEFFICIENTS = <real>c0 <real>c1 <real>c2
    SURFACE EFFECT = <string>NONE|SIMPLE_ON_OFF(NONE)
    FREE SURFACE EFFECT COEFFICIENTS = <real>coeff1
      <real>coeff2
  END [LAYER <string>layer_name]
  ACTIVE PERIODS = <string list>period_names
  INACTIVE PERIODS = <string list>period_names
END [CAVITY EXPANSION]
```

The `CAVITY EXPANSION` command block is used to apply a cavity expansion boundary condition to a surface on a body. This boundary condition is typically used for earth penetration studies where some type of projectile (penetrator) strikes a target. For a more detailed explanation of the numerical implementation of the cavity expansion boundary condition and the parameters for this boundary condition, consult Reference [1]. The cavity expansion boundary condition is a complex boundary condition with several options, and the detailed explanation of the implementation of the boundary condition in the above reference is required reading to fully understand the input parameters for this boundary condition.

There are two types of cavity expansion—cylindrical expansion and spherical expansion. Either the spherical or cylindrical option may be selected using the `EXPANSION RADIUS` command line; the default is `SPHERICAL`. Reference [1] describes these two types of cavity expansion.

The `surface set commands` portion of the `CAVITY EXPANSION` command block defines a set of surfaces associated with the cavity expansion pressure field. The surface specification commands work the same as for the pressure boundary condition as described in section 7.6.1.1.

The cavity expansion boundary condition generates a pressure at a node based on the velocity and surface geometry at the node. Since cavity expansion is essentially a pressure boundary condition, cavity expansion must be specified for a surface.

The target has a top free surface with a normal in the global positive z -direction; the target has a bottom free surface with a normal in the global negative z -direction. The point on the global z -axis intersected by the top free surface is given by the parameter `top_surface_zcoord` on the `FREE SURFACE` command line. The point on the global z -axis intersected by the bottom free surface is given by the parameter `bottom_surface_zcoord` on the `FREE SURFACE` command line.

It is necessary to define two points that lie on the axis (usually the axis of revolution) of the penetrator. These two nodes are specified with the `NODE SETS TO DEFINE BODY AXIS` command line. The first node should be a node toward the tip of the penetrator (`nodelist_1`), and the second node should be a node toward the back of the penetrator (`nodelist_2`). Only one node is allowed in each node set.

It is necessary to compute either a spherical or cylindrical radius for nodes on the surface where the cavity expansion boundary condition is applied. This is done automatically for most nodes. The calculations for these radii break down if the node is close to or at the tip of the axis of revolution of the penetrator. For nodes where the radii calculations break down, a user-defined radius can be specified with the `TIP RADIUS` command line. For more information, consult Reference [1].

Embedded within the target can be any number of layers. Each layer is defined with a `LAYER` command block. The command block begins with

```
BEGIN LAYER <string>layer_name
```

and is terminated with:

```
END [LAYER <string>layer_name]
```

Here the string `layer_name` is a user-selected name for the layer. This name must be unique to all other layer names defined in the `CAVITY EXPANSION` command blocks. The layer properties are defined by several different command lines—`LAYER SURFACE`, `PRESSURE COEFFICIENTS`, `SURFACE EFFECT`, and `FREE SURFACE EFFECT COEFFICIENTS`. These command lines are described next.

- `LAYER SURFACE = <real>top_layer_zcoord
<real>bottom_layer_zcoord`

The layer has a top surface with a normal in the global positive z -direction; the layer has a bottom surface with a normal in the global negative z -direction. In the `LAYER SURFACE` command line, the point on the global z -axis intersected by the top layer surface is given by the parameter `top_layer_zcoord`, and the point on the global z -axis intersected by the bottom layer surface is given by the parameter `bottom_layer_zcoord`.

- `PRESSURE COEFFICIENTS = <real>c0 <real>c1 <real>c2`

The value of the pressure at a node is derived from an equation that is quadratic based on some scalar value derived from the velocity vector at the node. The three coefficients for the

quadratic equation (c_0, c_1, c_2) in the `PRESSURE COEFFICIENTS` command line define the impact properties of a layer.

- `SURFACE EFFECT = <string>NONE|SIMPLE_ON\str (NONE)`
There can be no surface effects associated with a layer, or there can be a simple on/off surface effect model associated with a layer. The type of surface effect is determined by the `SURFACE EFFECT` command line. The default is no surface effects. If the `SIMPLE_ON_OFF` model is chosen, it is necessary to specify free surface effect coefficients with the `FREE SURFACE EFFECT COEFFICIENTS` command line.
- `FREE SURFACE EFFECT COEFFICIENTS = <real>coeff1 <real>coeff2`
All the parameters defined in a `LAYER` command block apply to that layer. If a simple on/off surface effect is applied to a layer, the surface effect coefficients are associated with the layer values. The surface effect parameter associated with the top of the layer is `coeff1`; the surface effect parameter associated with the bottom of the layer is `coeff2`.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines provide an additional option for cavity expansion. These command lines can activate or deactivate cavity expansion for certain time periods. See Section 2.6 for more information about these command lines.



Explicit Only

7.14.2 Silent Boundary

```
BEGIN SILENT BOUNDARY
  SURFACE = <string list>surface_names
  REMOVE SURFACE = <string list>surface_names
  ACTIVE PERIODS = <string list>period_names
  INACTIVE PERIODS = <string list>period_names
END [SILENT BOUNDARY]
```

The `SILENT BOUNDARY` command block is also called a non-reflecting surface boundary condition. A wave striking this surface is not reflected. This boundary condition is implemented with the techniques described in Reference [2]. The method described in this reference is excellent at transmitting the low- and medium-frequency content through the boundary. While the method does reflect some of the high-frequency content, the amount of energy reflected is usually minimal. On the whole, the silent boundary condition implemented in Presto is highly effective.

In the `SURFACE` command line, a series of surfaces may be listed through the string list `surface_names`. There must be at least one `SURFACE` command line in the command block. The `REMOVE SURFACE` command line allows the user to delete surfaces from the set specified in the `SURFACE` command line(s) through the string list `surface_names`. See Section 7.1.1 for more information about the use of these command lines for creating a set of surfaces used by the boundary condition.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines provide an additional option for the boundary condition. These command lines are used to activate or deactivate the boundary condition for certain time periods. See Section 2.6 for more information about these command lines.



Explicit Only

7.14.3 Spot Weld

```

BEGIN SPOT WELD
  NODE SET = <string list>nodelist_ids
  REMOVE NODE SET = <string list>nodelist_ids
  SURFACE = <string list>surface_ids
  REMOVE SURFACE = <string list>surface_ids
  SECOND SURFACE = <string>surface_id
  NORMAL DISPLACEMENT FUNCTION =
    <string>function_nor_disp
  NORMAL DISPLACEMENT SCALE FACTOR =
    <real>scale_nor_disp(1.0)
  TANGENTIAL DISPLACEMENT FUNCTION =
    <string>function_tang_disp
  TANGENTIAL DISPLACEMENT SCALE FACTOR =
    <real>scale_tang_disp(1.0)
  FAILURE ENVELOPE EXPONENT = <real>exponent(2.0)
  FAILURE FUNCTION = <string>fail_func_name
  FAILURE DECAY CYCLES = <integer>number_decay_cycles(10)
  SEARCH TOLERANCE = <real>search_tolerance
  IGNORE INITIAL OFFSET = NO|YES(NO)
  ACTIVE PERIODS = <string list>period_names
  INACTIVE PERIODS = <string list>period_names
  CREATE VISUALIZATION ELEMENTS = OFF|ON(OFF)
END [SPOT WELD]

```

The spot weld option lets the user model an “attachment” between a node on one surface and a face on another surface. This option can model a weld, small screw, or bolt.

The weld utilizes functions to represent either force or traction–displacement relations. Two force or traction–displacement curves are required for the spot weld model: one curve models normal behavior, while the other models tangential response. Behavioral differences between normal and tangential components necessitate separating the response components; the node may interpenetrate the surface, resulting in a negative normal displacement between the connection points. Therefore, the normal force displacement curve *must* have the ability to capture negative displacements (and forces). Although this penalty stiffness approach will work to prevent interpenetration, it is best to model this behavior using contact. Tangential displacement, however, is always positive.

Sample normal and tangential force–displacement curves are illustrated in Figures 7.8a and 7.8b, respectively. The displacements shown are the normal or tangential distances that the node moves from the nearest point on the face as measured in the original configuration. The corresponding forces shown are the forces at the attachment as functions of the distance between the two attachment points. The force–displacement curve assumes the two attachment points are originally at the same location and the initial distance and force are zero at time zero.

The sign of the normal displacement of the spot weld (generally tension at positive displacement or compression at negative displacement) is determined by the relative motion of the attached node and face as well as the normal direction of the face; see Figure 7.9. Relative motion of the face

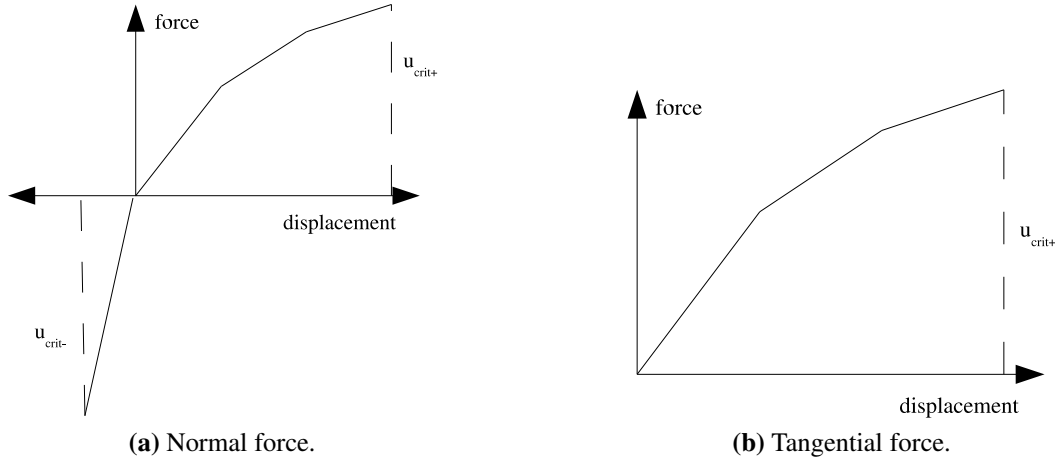


Figure 7.8: Spot weld force–displacement curves.

along the face normal causes compressive spot weld response, while relative motion of the face *opposite* to the face normal leads to tensile spot weld response.

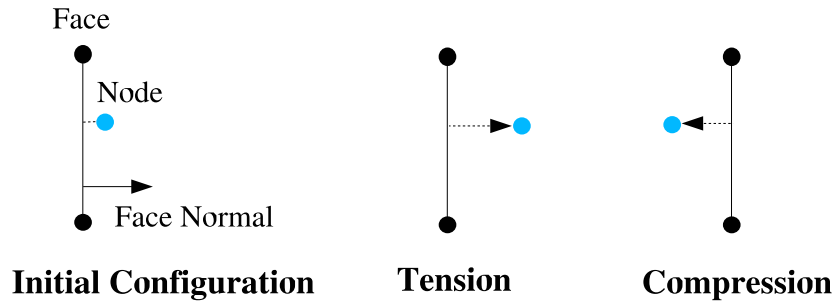


Figure 7.9: Sign convention for spot weld normal displacements.

The attachment in Presto is defined between a node on one surface and the closest point on an element face on the other surface. Since a face is used to define one of the attachment points, it is possible to compute a normal vector and a tangent vector associated with the face. This allows resolution of displacement (distance, both positive and negative) and force (both tensile and compressive) into normal and tangential components.

Note that the direction vector used to decompose normal and tangential deformations in the weld is the current normal of the face associated with the spot weld. See Figure 7.10 for a depiction of how normal and tangential displacements are computed in the spot weld. The spot weld deformation is decomposed into normal (SW_ND) and tangential (SW_TD) components.

With normal and tangential vectors associated with the attachment, the attachment can be characterized for the case of pure tension and pure shear. The pure tensile and pure shear response of the spot weld will be governed by the input force/traction–displacement functions. Presto includes two mechanisms for determining failure for cases that fall between pure tension and pure shear. In the first case, failure is governed by the equation

$$(u_n/u_{n_{crit}})^p + (u_t/u_{t_{crit}})^p < 1.0. \quad (7.6)$$

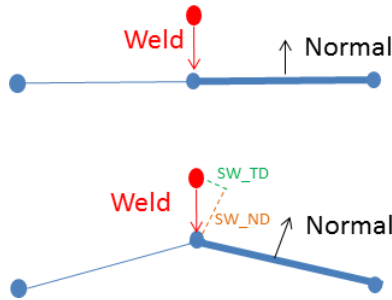


Figure 7.10: Spot weld normal and tangent.

In Equation (7.6), the distance from the node to the original attachment point on the face as measured normal to the face is u_n , which is defined as the normal distance. The maximum value given for u_n in the normal displacement curve is $u_{n_{crit}}$, but is different for positive and negative displacements. In Figure 7.8a, the value used for $u_{n_{crit}}$ is u_{crit+} in the positive direction and u_{crit-} in the negative direction. The distance from the node to the original attachment point on the face as measured along a tangent to the face is u_t , which is defined as the tangential distance. The maximum value given for u_t in the tangential displacement curve is $u_{t_{crit}}$. The value p is a user-specified exponent that controls the shape of the failure surface.

Alternatively, Presto permits a user-specified function to determine the failure surface. The function defines the ratio of $u_t/u_{t_{crit}}$ at which failure will occur as a function of $u_n/u_{n_{crit}}$. The function must range from 0.0 to 1.0, and have a value of 1.0 at 0.0 and a value of 0.0 at 1.0. These restrictions preserve proper failure for the cases of pure tension and pure shear.

To use the spot weld option in Presto, a `SPOT WELD` command block begins with the input line:

```
BEGIN SPOT WELD
```

and is terminated with the input line :

```
END [SPOT WELD]
```

A set of element faces which define one side of the spot weld (denoted the *first* surface) is specified with the `SURFACE` command line. The `SURFACE` command line can list one or more surfaces. Any surface listed on the `SURFACE` command line can be deleted from the list of surface by using a `REMOVE SURFACE` command line

When a spot weld is defined by a node set using a `NODE SET` command line and a surface the spot weld normal and tangential displacement functions define force/displacement relations. Alternatively a spot weld can be defined by a surface and a `SECOND SURFACE` command line. When a spot weld is defined by two surfaces the normal and tangential displacement functions define traction/displacement relations. A spot weld command block must include either a `NODE SET` command line or a `SECOND SURFACE` command line but not both. In either case the faces used for the spot weld constraints are always defined by the `SURFACE` command line.

For the `NODE SET` surface spot weld each node in `NODE SET` will look for a nearby face in `SURFACE`. If a sufficiently close face is found a spot weld will be created between the node and face using the force/displacement relations defined by the force and normal displacement curves. For the `SECOND SURFACE` surface spot weld each node in `SECOND SURFACE` will look for a nearby face in `SURFACE`. If a sufficiently close face is found a spot weld will be created between the node and the face. The node face relation will use a force/displacement relation defined by multiplying the weld displacement function (assumed to be a traction/displacement relation) by the tributary surface error of the node to compute a force/displacement function unique to that particular node. Thus the total max force resisted by a `NODE SET` based weld is the number of nodes in the set times the maximum value of the provided force/displacement curve. The total max force resisted by a `SECOND SURFACE` based weld is the area of the second surface times the maximum value of the provided traction/displacement curve.

The normal force/traction–displacement curve is specified by a function named by the string `function_nor_disp` in the `NORMAL DISPLACEMENT FUNCTION` command line. The last points in the positive and negative directions are used as the displacements beyond which the spot weld fails. This function can be scaled by the real value `scale_nor_disp` in the `NORMAL DISPLACEMENT SCALE FACTOR` command line. The scaling factor scales the y values (force/traction) of the normal displacement function. Thus a scaling factor of 2.0 would indicate that for a given displacement the spot weld has twice the force defined in `function_nor_disp`. The default scale factor is 1.0.

The tangential force/traction–displacement curve is specified by a function named by the string `function_tang_disp` in the `TANGENTIAL DISPLACEMENT FUNCTION` command line. The last point in the positive direction in the tangential curve is used as the displacement beyond which the spot weld fails. This function can be scaled by the real value `scale_tang_disp` given in the `TANGENTIAL DISPLACEMENT SCALE FACTOR` command line; the default for this factor is 1.0. As with the normal displacement functions the scaling factors apply to the y value (force/tractions) used by the spot weld.

The failure surface between pure tension and pure shear is controlled by specifying either the failure envelope exponent, p in Equation (7.6), or a failure function. The failure exponent is specified by the real value `exponent` in the `FAILURE ENVELOPE EXPONENT` command line. The failure function is specified by the `FAILURE FUNCTION` command line. If both a failure function and a failure exponent are given, then the failure function is used.

For an explicit transient dynamics code like Presto it is better to remove the force for the spot weld over several load steps rather than over a single load step once the failure criterion is exceeded. The `FAILURE DECAY CYCLES` command line controls the number of load steps over which the final force is removed (default value is 10). To remove the final force at a spot weld over five time steps, the integer specified by `number_decay_cycles` would be set to 5. Once the force at the spot weld is reduced to zero, it remains zero for all subsequent time (despite the function definition). That is, once a weld has reached the failure displacement, it will forever after be broken and will not reconnect if the node and face later move together again.

The user must set a tolerance for the node-to-face search with the

```
SEARCH TOLERANCE = <real>search_tolerance
```

command line. The value selected for `search_tolerance` will depend upon the distance between the nodes and surfaces used to define the spot weld.

If the user sets `IGNORE INITIAL OFFSET = YES`, the initial distance between the node and the face will be taken as the zero normal displacement distance for the spot weld. When using the `IGNORE INITIAL OFFSET = YES` command the initial normal force in the will be the value of the force-displacement curve at displacement=zero. The sign convention used to determine if a spot weld is in compression or tension is the same as without using `IGNORE INITIAL OFFSET = YES` as shown Figure 7.11.

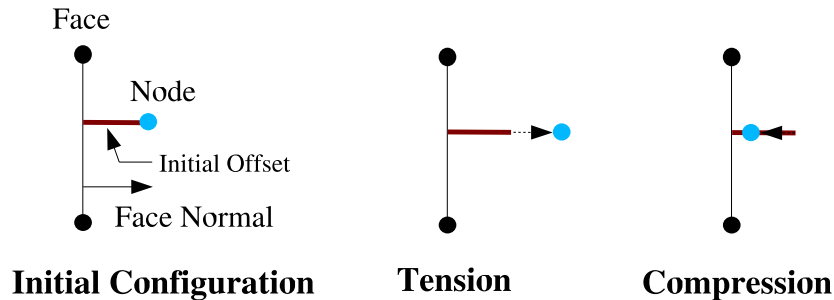


Figure 7.11: Sign convention for spot weld normal displacements with `IGNORE INITIAL OFFSET = YES`.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines are used to activate or deactivate the boundary condition for certain time periods. See Section 2.6 for more information.

The `CREATE VISUALIZATION ELEMENTS` command line triggers the creation of beam elements to visualize the spot weld in output. These elements are for visualization purposes only and have no effect on the spot weld behavior. The default value is `OFF`. Each beam element is drawn between the welded node and the corresponding contact point on the welded surface.

Some special output data can be obtained from spot welds. The list of available output variables is documented in Table 9.7.

7.14.3.1 Spot Weld Limitations and Usage Guidelines



Warning: In explicit dynamics, spot welds do not inherently have an *element* critical time step because they have zero mass. However, spot welds may still decrease the global system time step. When spot welds with high stiffness relative to their attached elements are used in the model, the default *element-based* critical time step estimate may lead to instability in the spot weld.

When a node-based time step estimate is used, spot welds contribute their stiffness to the nodes associated with the spot weld, thus influencing the nodal time step (see Section 3.2.3.1). Thus, the node-based time step should remain stable, even in the presence of a stiff spot weld.

The spot weld stiffness is calculated conservatively as the sum of the current derivatives of the normal and tangential force functions. If the nodes attached to an element that governs the criti-

cal time step have spot weld stiffness contributions, log file output will indicate the approximate percentage reduction in the critical time step due to the spot weld stiffness contribution.



Warning: If the offset between the spot weld node and the attached face becomes large, the weld may become unstable. This instability is related to potentially inconsistent handling of moments induced by weld forces acting over a finite gap. It is recommended that the normal gap offset between the spot weld node and the spot weld face not exceed roughly one-half of the characteristic element length in the neighboring mesh. Note that a large gap in either initial or deformed coordinates are both problematic. Thus, welds that can undergo large normal displacements prior to breakage may encounter stability problems once the normal gap in the weld is large.

If spot welds are used to represent bolts that are larger than the mesh size, it is recommended to use rigid patches on both sides of the interface. Rigid patches prevent concentrated forces and large deformations at the spot weld attachment point, such as in Figure 7.10. Spot welds attached to rigid patches will also exhibit enhanced stability, for instance, under mesh refinement, since the characteristic face size is bound by the rigid patch size rather than the size of an individual face.

7.14.3.2 Example

The following is an example of a basic spot weld defined between the nodes of `nodelist_17` and the faces of surface `surface_3`. Note the spot weld has a stiffness of $500e+2$ in compression, a stiffness of $100e+2$ in tension, and a stiffness of $50e+2$ in shear. Once either the normal or tangential displacement in the weld reaches a value of $1.0e-2$, the weld will fracture.

```
BEGIN FUNCTION spot_norm
  TYPE = PIECEWISE LINEAR
  BEGIN VALUES
    -1.0e-2 -500.0
    0.0      0.0
    1.0e-2  100.0
  END
END
```

```
BEGIN FUNCTION spot_tang
  TYPE = PIECEWISE LINEAR
  BEGIN VALUES
    0.0e-2  0.0
    1.0e-2  50.0
  END
END
```

```
BEGIN SPOT WELD
  NODE SET = nodelist_17
```

```

SURFACE = surface_3
NORMAL DISPLACEMENT FUNCTION = spot_norm
TANGENTIAL DISPLACEMENT FUNCTION = spot_tang
SEARCH TOLERANCE = 1.0e-3
END

```



7.14.4 Line Weld

```

BEGIN LINE WELD
  SURFACE = <string list> surface_names
  REMOVE SURFACE = <string list> surface_names
  BLOCK = <string list> block_names
  REMOVE BLOCK = <string list> block_names
  SEARCH TOLERANCE = <real>search_tolerance
  R DISPLACEMENT FUNCTION = <string>r_disp_function_name
  R DISPLACEMENT SCALE FACTOR = <real>r_disp_scale
  S DISPLACEMENT FUNCTION = <string>s_disp_function_name
  S DISPLACEMENT SCALE FACTOR = <real>s_disp_scale
  T DISPLACEMENT FUNCTION = <string>t_disp_function_name
  T DISPLACEMENT SCALE FACTOR = <real>t_disp_scale
  R ROTATION FUNCTION = <string>r_rotation_function_name
  R ROTATION SCALE FACTOR = <real>r_rotation_scale
  S ROTATION FUNCTION = <string>s_rotation_function_name
  S ROTATION SCALE FACTOR = <real>s_rotation_scale
  T ROTATION FUNCTION = <string>t_rotation_function_name
  T ROTATION SCALE FACTOR = <real>t_rotation_scale
  FAILURE ENVELOPE EXPONENT = <real>k(2.0)
  FAILURE DECAY CYCLES = <integer>number_decay_cycles
  ACTIVE PERIODS = <string list>period_names
  INACTIVE PERIODS = <string list>period_names
END LINE WELD

```

The line-weld capability is used to weld the edge of a shell to the face of another shell. The bond can transmit both translational and rotational forces. When failure of the line weld occurs, it breaks and no longer transmits any forces.

The edge of the shell that is tied to a surface is modeled with a block of one-dimensional elements (truss, beam, spring, etc.). The edge of the shell and the one-dimensional elements will share the same nodes. We will refer to the shell edge and the one-dimensional elements associated with it as the one-dimensional part of the line-weld model. The element blocks with the one-dimensional elements are specified by using the `BLOCK` command line. More than one element block can be listed on this command line. The element blocks referenced by the `BLOCK` command line must be one-dimensional elements—truss, beam, spring, etc.

The other part of the line weld is a set of faces defined by shell elements; this set of faces is the two-dimensional part of the line weld. The surface (the two-dimensional part of the model) to which the nodes (from the one-dimensional part of the model) are to be bonded is defined by any

surface of element faces derived from shell elements. The line weld will bond each node in the element blocks listed in the `BLOCK` command line to the closest face (or faces) of element faces in the surfaces listed in the `SURFACE` command line. More than one surface can be listed on this command line.

The command line `SEARCH TOLERANCE` sets a tolerance on the search for node-to-face interactions. For a given node, only those faces within the distance set by the `search_tolerance` parameter will be searched to determine whether the node should be welded to the face.

Each section of the line weld has its own local coordinate system (r, s, t) . The r -direction lies along a one-dimensional element (and hence on the surface). The s -direction lies on the surface and is tangential to the one-dimensional element. The t -direction lies normal to the face and is orthogonal to the r - and s -directions. Force-displacement functions and moment-rotation functions may be specified for all axes in the local coordinate system. If one of the functions is left out, the resistance is zero for that axis. These functions are similar to the ones used for the spot weld (see Figure 7.8b).

The force-displacement function in the r -direction represents shear resistance in the direction of the weld; this function is specified by a SIERRA function name on the `R DISPLACEMENT FUNCTION` command line. The force-displacement in the s -direction represents shear resistance tangential to the weld; this function is specified by a SIERRA function name on the `S DISPLACEMENT FUNCTION` command line. The force-displacement in the t -direction function represents tearing resistance normal to the surface; this function is specified by a SIERRA function name on the `T DISPLACEMENT FUNCTION` command line. The moment-rotation about the r -axis represents a rotational tearing resistance; this is specified by a SIERRA function name on the `R ROTATION FUNCTION` command line. The rotational resistances about the s -direction and the t -direction are likely not very meaningful, as rotations along these axes should be well constrained by the normal and tangential displacement relations. These two rotational resistances, if used, are defined with SIERRA function names on the `S ROTATION FUNCTION` and `T ROTATION FUNCTION` command lines. Note that each SIERRA function used in this command block is defined via a `DEFINITION FOR FUNCTION` command block in the SIERRA scope.

Any of the above functions can be scaled by using a corresponding scale factor. For example, the force-displacement function on the `R DISPLACEMENT FUNCTION` command line can be scaled by the parameter `r_disp_scale` on the `R DISPLACEMENT SCALE FACTOR` command line. Only the force values of the force-displacement curve will be scaled.

The failure function for the line weld is similar to that for the spot weld. Denote the displacement or rotation associated with a line weld as δ . Suppose that δ_i is a displacement in the r -direction. The force-displacement curve specified on the `R DISPLACEMENT FUNCTION` command line has a maximum value η . This is the maximum displacement the weld can endure in the r -direction before breaking. Associate this value of η with δ_i by designating it as η_i . Repeat this pairing process for all the displacements and rotations defining the line weld. Each displacement component in the line weld will be paired with one of the three maximum displacement values associated with the line weld. Each rotation component in the line weld will be paired with one of the three maximum rotation values associated with the line weld. Breaking of the weld under combined loading is calculated the same as the spot weld. The weld breaks under the following condition:

$$\sqrt[k]{\sum \left(\frac{\delta_i}{\eta_i} \right)^k} > 1. \quad (7.7)$$

In the above equation, the parameter k is set by the user. A typical value for k is 2. The summation takes place over all the failure functions (force-displacement and moment-rotation) for all the nodes. (The value for k is specified on the `FAILURE ENVELOPE EXPONENT` command line.)

For an explicit, transient dynamics code like Presto, it is better to remove the forces for the line weld over several time steps rather than over a single time step once the failure criterion is exceeded. The `FAILURE DECAY CYCLES` command line controls the number of time steps over which the final force is removed. To remove the final force at a line weld over five time steps, the integer specified by `number_decay_cycles` would be set to 5. Once the force in the line weld is reduced to zero, it remains zero for all subsequent time.

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines provide an additional option for the boundary condition. These command lines are used to activate or deactivate the boundary condition for certain time periods. See Section 2.6 for more information about these command lines.



7.14.5 Shell Solid Joint

```
BEGIN SHELL SOLID JOINT
  # block of shell elements
  BLOCK = <string list>block_names
  # Set of solid nodes
  NODE SET = <string list>nodelist_names
  SEARCH TOLERANCE = <real>tol
  TRUE GAUGE TEST = OFF|ON(OFF)  #EXPERIMENTAL DO NOT USE
  MASTER SELECTION = NORMAL PROJECTION|
    NEAREST NODE(NORMAL PROJECTION)
  FOLLOW THICKNESS CHANGE = OFF|ON (OFF)
  ROTATION UPDATE TYPE = REISSNER|KIRCHHOFF (REISSNER)
  MEAN QUADRATURE ROTATION = OFF|ON (OFF)
END [SHELL SOLID JOINT]
```

The shell solid joint capability connects a set of solid nodes to a shell element block. Typically it is used to connect the nodes on the surface of a high-fidelity solid element block to the edge of a low-fidelity shell element block.

A shell solid joint enforces that the translations of the solid nodes remain compatible with the shell translations and rotations at the interface. The solid nodes will conform to the shell kinematic assumptions (e.g., plane sections remain plane) as if embedded in the shell. The through-thickness thinning or expansion of the solid mesh in the joint region is constrained by the thickness change of the shells. The shell solid joint constraint equations are written at the solid nodes, i.e., they constrain each solid node within search tolerance to a single shell element.

The `BLOCK` command selects one or more sets of shell elements to constrain the solid nodes to. These must be two dimensional shell—not membrane—formulation elements. The `NODE SET`

command selects a set of nodes to constrain to the shells. These may be nodes connected to any type of solid element (i.e., they should not have rotational degrees of freedom).

The `SEARCH TOLERANCE` command optionally sets the tolerance for how close nodes need to be to shells to become constrained. If the `SEARCH TOLERANCE` command is not used a search tolerance is selected automatically based on the shell thickness, lofting factor, and in-plane size. The automatic search tolerances are set such that the joint will constrain any solid node found within or nearby the volumetric domain occupied by the theoretical shell material. A pictorial representation of how search tolerances are used is shown in Figure 7.12.

The `TRUE GAUGE TEST` option is experimental and should not be used.

The `MASTER SELECTION` command is used to change how the constrained nodes are projected to the plane of the shell. By default the constraint is made via a projection of the node along the shell normal onto the plane of the shell. The `NEAREST NODE` option will instead project the constrained node to the nearest node on the shell. The `NEAREST NODE` option can potentially be useful when the shell and solid meshes have been constructed contiguously such that a line of nodes on the solid side directly one to one maps to a single node on the shell side. For such a case use of the `NEAREST NODE` option will remove any possibility of mapping errors due to shell warping or curvature. However, the default `NORMAL PROJECTION` option should always be used for all general or non-matching shell and solid meshes.

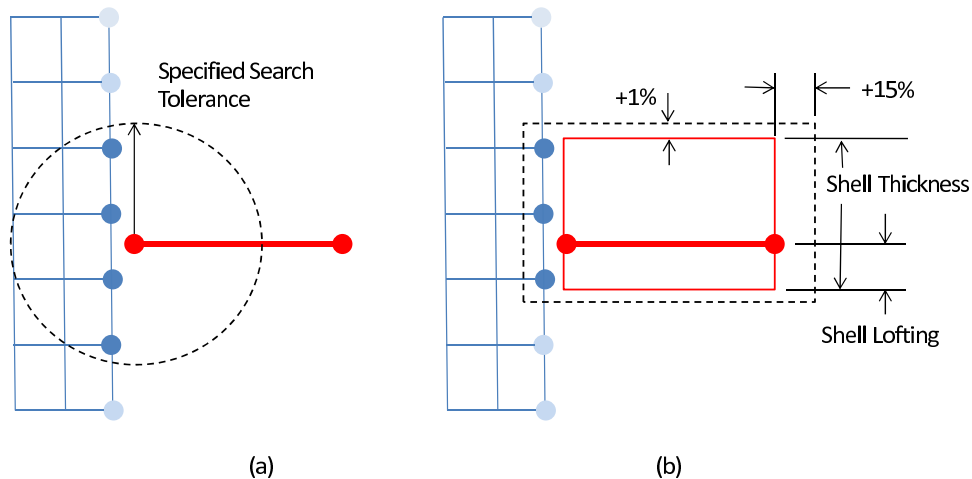


Figure 7.12: Search tolerances for shell-solid joints. (a) input; (b) automatic

7.14.5.1 Shell Solid Joint in Implicit Computations



Warning: Though the shell solid joint is available for implicit computations it is considered experimental for this use case. There are many known issues with the implicit shell solid joint. Convergence tends to be poor and serial analyses and potentially worse in parallel analyses.



Warning: In implicit the shell solid joint constraint is enforced through the control contact multilevel solver. Thus when using the shell solid joint in implicit a control contact solver block must be present in the solver input.

A number of implicit only options exist for the shell solid joint.

By default the movement of the solid nodes on the interface interface is uncoupled to any Poisson thinning or thickening of the attached shell. The `FOLLOW THICKNESS CHANGE = ON` option will turn on coupling of this degree of freedom ensuring that the solid nodes move up or down the shell normal consistently with the shell thickness changes. This option can potentially lead to a more accurate coupling of the joint during large membrane stretching. However, the coupling may also in some circumstances over constrain the joint causing robustness issues.

The `ROTATION UPDATE TYPE` command defines how shell solid joints computes coupling rotations. The default `REISSNER` option couples rotations to the rotational degrees of freedom of the shell nodes. The `KIRCHHOFF` option couples rotations to changes in shell element normal. Generally both options should yield the similar results. The `KIRCHHOFF` option could in some cases produces a stiffer but more stable result. This stability can be particular important if constraining a stiff material shell to a soft material solid.

The `MEAN QUADRATURE ROTATION` command allows tying rotations to the average shell rotation (`ON`) option rather than the nodal point (default `OFF`) option. This command has a similar effect to the `ROTATION UPDATE TYPE` where the `ON` option can create a potentially stiffer but more stable joint.

7.14.5.2 Restrictions and Assumptions

The following guidelines apply to both the explicit and implicit shell solid joint.

- Though the shell solid joint is a kinematic MPC, it can potentially artificially stiffen the nearby elements and thus it may be necessary to manually reduce the analysis time step a small amount to ensure integration stability.
- It is generally assumed that there are minimal or no lateral gaps or overlaps in the joint. Gaps or overlaps between the shell block(s) and solid(s) will tend to introduce additional artificial stiffness into the joint.
- Any offset between the solid cross-section neutral bending plane and the shell midplane will result in an effective lofting distance, which may increase the stiffness of the structure unexpectedly. (This is because the enforcement of shell kinematics on the solid nodes forces the solid to rotate and bend around the shell midplane as if part of the shell.)
- In order to correctly enforce moment and rotation compatibility across the joint, the joint must contain at least two solid nodes in the shell normal (i.e., thickness) direction. It is preferable to have five or more solid nodes in the shell normal direction for a smoothly connected joint.
- For best results the solid side of the joint should be meshed as fine or finer than the shell side. A finely meshed shell connected to a coarsely meshed solid will result in constraints at only the solid nodes, leading to unconstrained shells nodes at the joint.
- The shell solid joint constraint can constrain each solid node to only one shell. In ambiguous



- cases the “best” matching shell will be selected based on proximity.
- Care should be taken if using the shell solid joint on the same part of a mesh as contact or other kinematic or rigid body constraints. If multiple constraint types come into conflict generally either one of the constraints sets will not be enforced or stability problems will result.

7.14.6 Viscous Damping

```
BEGIN VISCOUS DAMPING <string>damp_name
#
# block set commands
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
#
MASS DAMPING COEFFICIENT = <real>mass_damping
ROTATIONAL MASS DAMPING COEFFICIENT = <real>rot_mass_damping
STIFFNESS DAMPING COEFFICIENT = <real>stiff_damping
VELOCITY DAMPING COEFFICIENT = <real>damping_coefficient
RIGID BODY INVARIANCE = OFF|ON(OFF)
#
# additional commands
ACTIVE PERIODS = <string list>period names
INACTIVE PERIODS = <string list>period_names
END [VISCOUS DAMPING <string>damp_name]
```

The `VISCOUS DAMPING` command block adds simple Rayleigh viscous damping to mesh nodes. At each node, Presto computes a damping coefficient, which is then multiplied by the node velocity to create a damping force. The damping coefficient is the sum of the mass times a mass damping coefficient and the nodal stiffness times a stiffness damping coefficient. In general, the mass damping portion damps out low-frequency modes in the mesh, while the stiffness damping portion damps out higher-frequency terms. Appropriate values for the damping coefficients depend on the frequencies of interest in the mesh. The general expression for the critical damping fraction, c_d , for a given frequency is

$$c_d = \frac{1}{2} \left(k_d \omega + \frac{m_d}{\omega} \right), \quad (7.8)$$

where k_d is the stiffness damping coefficient, m_d is the mass damping coefficient, and ω is the frequency of interest. The stiffness damping portion must be used with caution. Because this term depends on the stiffness, it can affect the critical time step. Thus certain ranges of values for the stiffness damping coefficient can change the critical time step for the mesh. As Presto does not currently modify the critical time step based on the selected values for this coefficient, some choices for this parameter can cause solution instability.

Velocity damping, on the other hand, is applied *directly* to the velocity update, i.e.,

$$\mathbf{v}_{new} = (1.0 - v_d) \mathbf{v}_{old}, \quad (7.9)$$

where v_d is the velocity damping coefficient and $\mathbf{v}$ represents the velocity.

7.14.6.1 Block Set Commands

The `block set commands` portion of the `VISCOUS DAMPING` command block defines a set of element blocks associated with the viscous damping and can include some combination of the following command lines:

```
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_names
```

These command lines provide a set of Boolean operators for constructing a set of element blocks. See Section 7.1.1 for more information about the use of these command lines for creating a set of element blocks used by viscous damping. There must be at least one `BLOCK` or `INCLUDE ALL BLOCKS` command line in the command block. All the nodes associated with the elements specified by the block set commands will have viscous damping forces applied.

7.14.6.2 Viscous Damping Coefficient

The mass damping coefficient, m_d , in Equation (7.8) is specified using the parameter `mass_damping` on the command line:

```
MASS DAMPING COEFFICIENT = <real>mass_damping
ROTATIONAL MASS DAMPING COEFFICIENT = <real>rot_mass_damping
```

Note, the `MASS DAMPING COEFFICIENT` affects only translational degrees of freedom while the `ROTATIONAL MASS DAMPING COEFFICIENT` affects only the rotational degrees of freedom on shells, beams, and rigid bodies. Mass damping most strongly damps the low-frequency modes, and in Presto, is applied as a boundary condition which does not affect the critical time step.

The stiffness damping coefficient, k_d , in Equation (7.8), is specified as the parameter `stiff_damping` on the command line:

```
STIFFNESS DAMPING COEFFICIENT = <real>stiff_damping
```

Stiffness damping most strongly damps high-frequency modes and is applied through a stress correction of the pressure term at the element level. Because this type of damping is done at the element level, it has a direct effect on the critical time step as seen in Equation (7.10), where $\epsilon = \frac{k_d \omega}{2}$ in the absence of mass damping and $\frac{2}{\omega} \approx \frac{m}{\kappa}$.

$$\Delta t \leq \frac{2}{\omega} \left(\sqrt{1 + \epsilon^2} - \epsilon \right) \quad (7.10)$$

The velocity damping coefficient, v_d , in Equation (7.9), is specified as the parameter `vel_damping` on the command line:

```
VELOCITY DAMPING COEFFICIENT = <real>vel_damping
```

Velocity damping applies a constant damping term that directly lowers velocity and damps out all modes. Velocity damping is primarily used to drive a dynamic solution towards static equilibrium during or after a preload procedure.

Velocity damping and mass damping *may not* be used in combination.

```
RIGID BODY INVARIANCE = OFF|ON(OFF)
```

By default mass and velocity damping apply a force to each node that opposes the current velocity of that node. This acts as if the part is moving through a viscous fluid. This force will damp out both vibration and net rigid body motions. The rigid body invariance option applies damping forces based on the difference between the node velocity and the net average velocity of the node set. The intent is to damp out vibrations in the part but not affect the net rigid body motions. Note, this option only makes sense if the damping is applied on somewhat contiguous parts. For example applying rigid body invariant damping to two pieces moving in opposite directions would still slow both parts as the net rigid motion of the parts moving away from one another is zero. An addition effect of the rigid body invariance on option is that the net force and moment applied by viscous damping will be assured to be zero.

7.14.6.3 Additional Command

The ACTIVE PERIODS and INACTIVE PERIODS command lines can optionally appear in the VISCOUS DAMPING command block:

```
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

This command line can activate or deactivate the viscous damping for certain time periods. See Section 2.6 for more information about this command line.

7.14.7 Artificial Strain

```
BEGIN ARTIFICIAL STRAIN
#
# Zone of application commands
INCLUDE ALL BLOCKS
ELEMENT = <int list>elem_nums
BLOCK = <string list>block_names
REMOVE BLOCK = <string list>block_names
#
# Strain definition commands
STRAIN TYPE = LOG|ENGINEERING(ENGINEERING)
FUNCTION = <string>function_name
    [SCALE FACTOR = <real>scale]
DIRECTION <string>dir_name FUNCTION = <string>dir_func
    [SCALE FACTOR = <real>scale]
DIRECTION FIELD <string>field_name FUNCTION =
```

```

    <string>field_dir_func [SCALE FACTOR = <real>scale]
R FUNCTION = <string>r_func [SCALE FACTOR = <real>scale]
S FUNCTION = <string>s_func [SCALE FACTOR = <real>scale]
T FUNCTION = <string>t_func [SCALE FACTOR = <real>scale]
END [ARTIFICIAL STRAIN]

```

A The `ARTIFICIAL STRAIN` command block defines an artificial strain applied on a block by block or element by element basis. Artificial strains may be useful for fitting parts together for a preload analysis or modeling a variety of other physics that could cause material expansion or contraction (such as radiation induced expansion, material desiccation, etc.)

Artificial strains may also be defined in the material block, see Section 5.1.3. However, defining artificial strains by boundary condition as shown in this section is more flexible.

A variety of commands can be used to describe the set of elements this command block applies to as detailed in Section 7.1.1.

Artificial strains may be applied based in either the `LOG` or `ENGINEERING` strain space as defined from the `STRAIN TYPE` command. There are several ways to define the applied strain tensor. One and only one of the following command sets may be used.

- `FUNCTION` is a single function defining the isotropic strain to apply. The function is a time history function optionally scaled by an input constant.
- `DIRECTION` may be used to define up to three times to define up to three orthogonal directions paired with separate time history functions in which to apply strain. The direction names used here are defined in a Sierra scope `DIRECTION` command. The direction command is generally only applicable to three dimensional elements and defines the strain directions in the global XYZ coordinate system. The functions may be optionally scaled by an input constant.
- `DIRECTION FIELD` may be used along with `DIRECTION` lines to define up to three orthogonal directions paired with separate time history functions in which to apply strain. The field names given to `DIRECTION FIELD` refer to name of an element direction variable. The element direction variable must be a defined element output field and must have exactly three components at each element. The direction field may be populated by a number of methods including `USER OUTPUT` to define the field on the fly or use of `INITIAL CONDITION` blocks to read the field from the mesh file. Note, though the field used can potentially vary in time the behavior of artificial strain is not well defined for time variant directions and it is recommended that only directions that are constant in time be used. Additionally at each and every element the set of up to three `DIRECTION FIELD` directions and `DIRECTION` directions must be orthogonal to each other. The functions may be optionally scaled by an input constant.
- the `R`, `S`, `T FUNCTION` commands define strain in the element local RST coordinate system. For solids this generally corresponds to the global XYZ coordinate system. For shells and membranes `R` and `S` are the in plane directions and `T` is the through thickness direction.

For beams and trusses R is the axial direction and S and T are the lateral directions. These functions are time history functions and may be optionally scaled by an input constant.

Note, the artificial strain is not an absolute constraint. An element could be artificially shrunk and then stretched by actual material strain in a way that the element shape doesn't change, but the element does accumulate stress and strain energy. If artificial strain is applied uniformly to an unconstrained body that strain will grow or shrink the body as described by the artificial strain in a nominally stress free manner.

7.14.7.1 Restrictions and Usage Guidelines

Artificial strain cannot be simultaneously defined in a material and in the artificial strain block for the same element. Thermal strain may be defined in a material and artificial strain in an artificial strain block. However, in such cases the strain must be of matching type (LOG or ENGINEERING.)

One common error is assuming strain applied in the direction (1, 1, 0) has the same effect of applying an equal R and S strain. Applying a strain in the direction of 1, 1, 0 expands the body uniformly along the 45 degree direction vector. Applying a strain in both the R and S directions expands the body uniformly in the RS plane.

It is possible to apply any arbitrary strain state by computing the eigenvectors and values of the strain state and then applying direction strains in the directions of the eigenvectors with the magnitudes of the eigenvalues.

Note, strains should generally be applied by functions that are smooth in time. A sharp jump in the target artificial strain over a time step will tend to produce a strong shock in the system.



Explicit Only

7.14.8 Volume Repulsion Old

```
BEGIN VOLUME REPULSION OLD <string>repulsion
  FRICTION COEFFICIENT = <real>fric_coeff
  SCALE FACTOR = <real>scale_factor
  OVERLAP TYPE = [NODAL|VOLUMETRIC]

BEGIN BLOCK SET <string>set
  BLOCK = <string list>block_names
  INCLUDE ALL BLOCKS
  REMOVE BLOCK = <string list>block_names
  #
  SURFACE = <string list>surface_names
  REMOVE SURFACE = <string list>surface_names
  #
  ACTIVE PERIODS = <string list>period_names
  INACTIVE PERIODS = <string list>period_names
  #
  LINE CYLINDER RADIUS = <real>cylinder_radius
  ELEMENT REPRESENTATION = [BEAM_ELEMENT_CYLINDERS|
    TRUE_SOLID_VOLUME|NODES]
```

```

END [BLOCK SET <string>set]
END [VOLUME REPULSION OLD <string>repulsion]

```

The `VOLUME REPULSION OLD` command block is used to create a cylindrical volume around beam elements that is used as a frictional contact surface. Because beam elements are represented as line elements, there is no volume associated with the elements that can be used for contact. This command will generate nodal forces based on interpenetrations of a prescribed set of nodes with the cylindrical contact surfaces. The force is generated using a penalty stiffness method where the magnitude of the force depends on the mass of the node and the current time step. This command will be replaced by Dash contact in the future.

Definition of the coefficient of friction for the cylindrical surface is defined through the `FRICTION COEFFICIENT` command line.

The `SCALE FACTOR` command line specifies a scale factor that scales the force produced from node-surface interactions.

The `OVERLAP TYPE` command line must always be set to `VOLUMETRIC`.

7.14.8.1 Block Set

Complete definition of the contact surfaces and node sets are completely defined within multiple `BLOCK SET` command blocks. At least two `BLOCK SET` command blocks must be defined, one that defines the beam element blocks to wrap in cylinders by setting the `ELEMENT REPRESENTATION` command line to `BEAM_ELEMENT_CYLINDERS`. The second `BLOCK SET` command block defines the node set used to contact the cylindrical surfaces by setting the `ELEMENT REPRESENTATION` to `NODES`.

The `BLOCK`, `INCLUDE ALL BLOCKS` and `REMOVE BLOCK` command lines must be used when defining the beam element blocks, while a combination of these block command lines and the following surface command lines, `SURFACE` and `REMOVE SURFACE`, must be used to define the node set used for contact.

`ACTIVE PERIODS` command line defined the time periods in which this boundary condition is active, whereas the `INACTIVE PERIODS` command line defines when this boundary condition is inactive.

The `LINE CYLINDER RADIUS` command line specifies the radius of the cylindrical surface around the beam elements.

7.14.9 Tied Multi-Point Constraints

```

BEGIN TIED MPC
#
TIED FACES = <string list>BLOCKS/SURFACES
TIED NODES = <string list>BLOCKS/SURFACES/NODE SETS
#
# Tied contact search commands
SEARCH TOLERANCE = AUTO / <real> tolerance (AUTO)
VOLUMETRIC SEARCH TOLERANCE = <real>vtolerance

```

```

SEARCH CONFIGURATION = MODEL|CURRENT (CURRENT)
#
# DOF subset selection
COMPONENTS = <enum>X|Y|Z|RX|RY|RZ
REMOVE GAPS AND OVERLAPS = OFF|ON(OFF)
END [TIED MPC]

```

Sierra/SM provides a tied contact multi-point constraint (MPC) capability that allows a code user to tie a set of nodes to a set of faces. For tied contact MPCs, a proximity search is performed to create a set of MPCs that act as tied contact constraints. If the MPCs are created in this way, the search is performed at the time of initialization to find pairings of nodes (defined from the `TIED NODES` command line) to faces (defined from the `TIED FACES` command line). `TIED FACES` may be names of blocks and/or surfaces and `TIED NODES` may be names of blocks, surfaces and/or node sets. A separate constraint is created for each node in the `TIED NODES`. This type of MPC is equivalent to using pure master/slave tied contact and is significantly faster than standard tied contact for explicit dynamics analyses.

```

SEARCH TOLERANCE = AUTO / <real> tolerance (AUTO)

```

The `SEARCH TOLERANCE` command line is used to specify the maximum distance between a node and a face to create an MPC. This has a similar meaning to the search tolerance used in standard tied contact. By default, the search tolerance is set to `AUTO` which automatically computes the search tolerance value internally. The value that is computed as the search tolerance is 15 percent of the second smallest edge length of the face.

```

VOLUMETRIC SEARCH TOLERANCE = <real>vtolerance

```

In addition to creating node/face constraints, a search can be used to create volumetric constraints, in which a node is constrained to a volume bounded by faces. The `VOLUMETRIC SEARCH TOLERANCE` command is used to enable volumetric constraints and sets the tolerance used for the search. The `TIED NODES` are constrained to all nodes on the `TIED FACES` that are within the volumetric search tolerance, `vtolerance`.

Currently the primary usage of volumetric constraints is to constrain a meshed void placed inside another mesh. The volumetric search is used in conjunction with the node/face search. A set of node/face constraints is first created for the `TIED NODES` within the standard search tolerance of any `TIED FACES`. Next, a volumetric constraint is created for any remaining unconstrained nodes within the volumetric search tolerance of the node/face constrained nodes. The volumetric constraint is formulated in a way that approximates an isoparametric map.

```

SEARCH CONFIGURATION = MODEL|CURRENT (CURRENT)

```

The configuration in which nodes are searched for to create the MPC's can be changed via the `SEARCH CONFIGURATION` command. This can be useful if a restart or procedural transfer is performed and the MPC's need to change or remain the same as the start of the simulation.


```
COMPONENTS = <enum>X|Y|Z|RX|RY|RZ
```

By default, tied multi-point constraints are applied to all degrees of freedom of the nodes involved. For nodes that only have translational degrees of freedom, all three components (X, Y and Z) are constrained. Likewise, for nodes that have both translational and rotational degrees of freedom, all six components (X, Y, Z, RX, RY and RZ) are constrained. The `COMPONENTS` command line can be used to enforce the constraint on a subset of the degrees of freedom. If the `COMPONENTS` command line is included in a `TIED MPC` command block, only the components listed would be constrained.

To use tied MPCs, the `TIED FACES` and `TIED NODES` must be defined. It is important to note that the `TIED FACES` cannot contain any node sets.

The following example demonstrates how to use Tied MPCs between two surfaces:

```
BEGIN TIED MPC
  TIED FACES = surface_10
  TIED NODES = surface_11
END MPC
```

The following example demonstrates the usage of Tied MPCs for both tied and volumetric constraints. Assuming that `surface_10` surrounds the exterior surface of `block_1`, this block would result in tied contact between the nodes on the exterior of `block_1` and `surface_10`, and volumetric constraints between the nodes on the interior of `block_1` and `surface_1`.

```
BEGIN TIED MPC
  TIED FACES = surface_10
  TIED NODES = block_1
  VOLUMETRIC SEARCH TOLERANCE = 3.0
END MPC
```

Tied MPCs can potentially be in conflict with other constraints. Refer to [Appendix D](#) for information on how conflicting constraints are handled.

7.14.9.1 Resolve Multiple MPCs

The behavior of tied multi-point constraints is ill-defined when a node within the `TIED FACES` is constrained to more than one set of nodes within `TIED NODES`. Sierra/SM's tied MPC capability can handle chained MPCs, where a node in the `TIED FACES` is a node within the `TIED NODES` in another constraint, but it cannot simultaneously enforce multiple tied MPCs that have the same `TIED FACES` node.

```
RESOLVE MULTIPLE MPCS = ERROR|FIRST WINS|LAST WINS|ENFORCE ALL|
  IGNORE(ERROR)
```

The `RESOLVE MULTIPLE MPCS` command line, used within the region scope, controls how to resolve cases where a node within the `TIED NODES` is constrained to more than one set of nodes within the `TIED FACES`. Although multiple MPCs cannot be simultaneously enforced, this command provides ways to work around this problem that may be acceptable in many situations. The default option is `ERROR`, which results in an error message and terminates the code if this occurs. Alternatively, this command can take the following options:

FIRST WINS Keep only the first MPC seen at a node (MPC order generally determined by input deck order).

LAST WINS Keep only the last MPC seen at a node (MPC order generally determined by input deck order).

ENFORCE ALL Attempt to simultaneously enforce all MPCs seen at a node. As long as the MPCs are orthogonal this may work. If the MPCs are not orthogonal this will likely lead to an erroneous result or stability issues.

IGNORE If a node is found with multiple MPCs, ignore all of the conflicting MPCs and enforce none.

7.14.9.2 Remove Gaps and Overlaps

Depending on how the model is constructed, there may be a gap between the `TIED FACES` and the `TIED NODES` in the initial configuration. Tied MPCs are best posed when this gap is zero. The option `REMOVE GAPS AND OVERLAPS`, if set to `ON`, moves the MPC'd node to the constraint point during initialization, explicitly zeroing the gap. The default value for this command is `OFF`.

7.14.10 Master Slave Multi-Point Constraints

```
BEGIN MASTER SLAVE MPC
#
# Master/Slave MPC commands
MASTER NODE SET = <string list>master_nset
MASTER NODES = <integer list>master_nodes
MASTER WEIGHTS = <real list>master_weights
MASTER SURFACE = <string list>master_surf
MASTER BLOCK = <string list>master_block
SLAVE NODE SET = <string list>slave_nset
SLAVE NODES = <integer list>slave_nodes
SLAVE SURFACE = <string list>slave_surf
SLAVE BLOCK = <string list>slave_block
#
# DOF subset selection
COMPONENTS = <enum>X|Y|Z|RX|RY|RZ
END [MASTER SLAVE MPC]

# Control handling of multiple MPCs
RESOLVE MULTIPLE MPCS = ERROR|FIRST WINS|LAST WINS|
```

Sierra/SM provides a master slave multi-point constraint (MPC) capability that allows a code user to specify arbitrary constraints between sets of nodes. The commands to define a master/slave MPC are all listed within a `MASTER SLAVE MPC` command block. The master/slave type of MPC imposes a constraint between a set of master nodes and a set of slave nodes. The motion of the the slave nodes is constrained to be equal to the average motion of the master nodes. This type of MPC is typically most useful if there is either a single master node and one or more slave nodes, or multiple master nodes and a single slave node. If there are multiple slave nodes, they are constrained to move together as a set.

The sets of master and slave nodes used in the MPC can be defined by using a node set on the mesh file, a list of nodes provided in the input file, or a surface on the mesh file from which a list of nodes is extracted. This can be done for the set of master nodes using one or more of the following commands:

```
MASTER NODE SET = <string list>master_nset
MASTER NODES = <integer list>master_nodes
MASTER SURFACE = <string list>master_surf
MASTER BLOCK = <string list>master_block
MASTER WEIGHTS = <real list>master weights
SLAVE NODE SET = <string list>slave_nset
SLAVE NODES = <integer list>slave_nodes
SLAVE SURFACE = <string list>slave_surf
SLAVE BLOCK = <string list>slave_block
```

The `MASTER NODE SET` and `SLAVE NODE SET` command lines specify the names of node sets in the mesh file. The nodes in these node sets are included in the sets of master or slave nodes for the constraint.

The `MASTER NODES` and `SLAVE NODES` command lines specify lists of integer IDs of nodes to be included in the sets of master or slave nodes for the constraint.

The `MASTER SURFACE` and `SLAVE SURFACE` command lines specify the name of a surface in the mesh file. The nodes contained in this surface are included in the sets of master or slave nodes for the constraint.

The `MASTER BLOCK` and `SLAVE BLOCK` command lines specify the names of a block in the mesh file. The nodes contained in these blocks are included in the sets of master or slave nodes for the constraint.

The `MASTER WEIGHTS` command can be used to explicitly define a constraint equation. This command is subject to several restrictions. The `MASTER WEIGHTS` command must be used with a `MASTER NODES` command and cannot be used with a `MASTER NODE SET`, `MASTER SURFACE`, or `MASTER BLOCK` command. If the `MASTER WEIGHTS` is used the number number of weights must be the same as the number of nodes in the `MASTER NODES` command. Negative weights are allowed. However, negative MPC weights can potentially cause stability problems in explicit analysis.

```
COMPONENTS = <enum>X|Y|Z|RX|RY|RZ
```

By default, multi-point constraints are applied to all degrees of freedom of the nodes involved. For nodes that only have translational degrees of freedom, all three components (X, Y and Z) are constrained. Likewise, for nodes that have both translational and rotational degrees of freedom, all six components (X, Y, Z, RX, RY and RZ) are constrained. The `COMPONENTS` command line can be used to enforce the constraint on a subset of the degrees of freedom. If the `COMPONENTS` command line is included in a `MASTER SLAVE MPC` command block, only the components listed would be constrained.

The following is an example of use of the `MASTER WEIGHTS` command to define a Master Slave MPC.

```
BEGIN MASTER SLAVE MPC
  SLAVE NODES      = 23
  MASTER NODES     = 26      12      16
  MASTER WEIGHTS   = 0.5 0.25 0.25
  COMPONENTS       = X
END MPC
```

The above MPC adds a constraint that the X displacement at node 23 will always be equal to 0.5 times the X displacement at node 26 plus 0.25 the X displacement at node 12 plus 0.25 the X displacement at node 16.

Master Slave MPCs can potentially be in conflict with other constraints. Refer to [Appendix D](#) for information on how conflicting constraints are handled.

7.14.10.1 Resolve Multiple MPCs

The behavior of multi-point constraints is ill-defined when a master node is constrained to more than one set of slave nodes. Sierra/SM's MPC capability can handle chained MPCs, where a master node is a slave node in another constraint, but it cannot simultaneously enforce multiple MPCs that have the same slave node.

Refer to [Section 7.14.10.1](#) for documentation of the `RESOLVE MULTIPLE MPCs` command line.

7.14.11 Submodel

```
BEGIN SUBMODEL
  #
  EMBEDDED BLOCKS = <string list>embedded_block
  ENCLOSING BLOCKS = <string list>enclosing_block
END [SUBMODEL]
```

Sierra/SM provides a method to embed a submodel in a larger finite element model. The element blocks for both the submodel and the larger system model should exist in the same mesh file. The space occupied by the embedded blocks should also be occupied by the enclosing blocks.

This capability ties each node of the submodel to an element in the larger finite element model. The code makes no correction for mass due to volume overlap. However, this correction often can be done easily by hand simply by adjusting the density of the submodel block so that it is the difference between the density of the submodel block and the enclosing block.

The embedded blocks (the submodel blocks) and the enclosing blocks (the system model blocks) are specified using the following two line commands:

```
EMBEDDED BLOCKS = <string list>embedded_block
ENCLOSING BLOCKS = <string list>enclosing_block
```

For example, to embed `block_7` and `block_8` inside a system model where the embedded blocks are within `block_2`, `block_3`, and `block_5`, the following can be used:

```
BEGIN SUBMODEL
  EMBEDDED BLOCKS = block_7 block_8
  ENCLOSING BLOCKS = block2 block_3 block_5
END
```

7.14.12 RBE3

```
BEGIN RBE3
  #
  REFERENCE BLOCK|SURFACE|NODE SET|NODES = <string list>NAMES
  CONNECTION BLOCK|SURFACE|NODE SET|NODES = <string list>NAMES
  #
  # RBE3 WEIGHTS of 6 DOF's
  WEIGHTS = <double list(6)> X Y Z RX RY RZ
  #
  # Control of reference node mass distribution
  DISTRIBUTE REFERENCE NODE MASS = TRUE|FALSE(TRUE)
END [RBE3]
```

Sierra/SM provides the ability to define a rigid body element, or so-called RBE3, constraint between sets of nodes in a model. In the RBE3 formulation a set of ‘connection’ nodes and a set of ‘reference’ nodes are specified. Reference nodes are specified using one of the `REFERENCE BLOCK`, `REFERENCE SURFACE`, `REFERENCE NODE SET`, or `REFERENCE NODES` commands, while connection nodes can similarly be specified using one of `CONNECTION BLOCK`, `CONNECTION SURFACE`, `CONNECTION NODE SET`, or `CONNECTION NODES`. The RBE3 constraint prescribes a weighted average of the motion of the reference nodes to the connection nodes; currently, a mass-weighting scheme is used. Note that RBE3’s are *not* actually rigid elements. If used correctly, RBE3’s should not add stiffness to the system.

RBE3 constraints in Sierra/SM currently only provide a force-type coupling: forces and moments prescribed at the reference nodes are applied as equivalent forces on the connection nodes.

```
WEIGHTS = <double list(6)> X Y Z RX RY RZ
```

The `WEIGHTS` command line is used to specify the coupling weight applied to each corresponding force and moment component. For example, specifying `WEIGHTS = 1.0 1.0 1.0 1.0 0.0 0.0` will apply equivalent forces to the connection nodes computed from the x -, y -, and z -components of the forces and the x -component of the moments at the reference nodes (i.e., excluding the y - and z -components of the nodal moments from the constraint). By default, all six component weights are set to a value of 1.0.

The following example demonstrates how to use an RBE3 to tie two blocks:

```
BEGIN RBE3
  REFERENCE BLOCK = BLOCK_1
  CONNECTION BLOCK = BLOCK_2
  WEIGHTS = 1 1 1 1 1 1
END RBE3
```

The `DISTRIBUTE REFERENCE NODE MASS` command specifies whether the total mass of the reference nodes is used to update the accelerations—and, consequently, the forces—of the connection nodes. If this option is `FALSE`, the only effective masses $\tilde{m}_{ci}$ contributing to the accelerations of the connection nodes are the standard nodal masses m_{ci} . If this option is `TRUE`, the effective masses of the connection nodes are computed as

$$\tilde{m}_{ci} = m_{ci} + \sum_j w_{ji} m_{rj}$$

where m_{rj} are the reference node masses, and w_{ji} mass-weighting factors for distribution (the same weighting used for the RBE3 force and moment distribution). Note that when distribution of the reference node mass is enabled, the effective connection node masses $\tilde{m}_{ci}$ are only factored into the accelerations of the connection nodes; the actual nodal masses m_{ci} are not altered. `TRUE` is the default setting.



Warning: RBE3's do *not* currently transmit kinematic boundary conditions.



Warning: The reference node positions of the RBE3 are currently not updated with respect to the connection nodes.



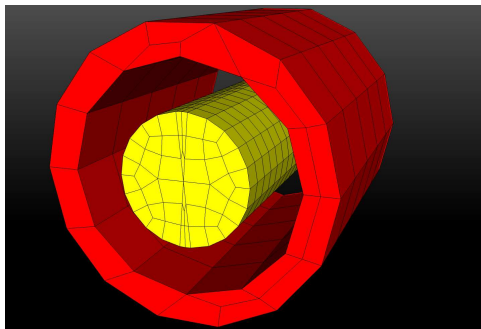
Known Issue: The RBE3 constraint is currently incompatible with mesh rebalance.

7.14.12.1 Sample RBE3 Usage

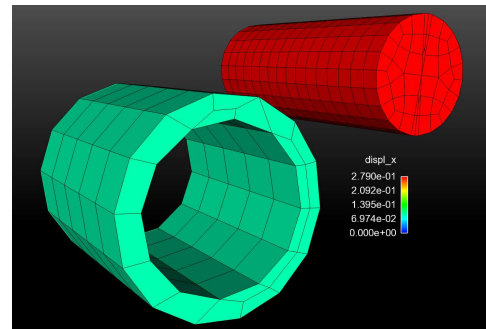
The following example illustrates how the RBE3 is capable of transferring only the x -component of any forces applied to the reference block. The problem consists of a cylinder at the center of a

pipe that has a larger inner radius than the cylinder's outer radius such that there is no contact. The problem setup is shown in Figure 7.13a. In this example, the cylinder is the reference block and the pipe is the connection block in the RBE3 input command block. A prescribed force is applied to the cylinder in the x -, y -, and z -directions, while a distribution weight is specified only in the x -direction, as shown below. Thus, the connection block transfers only the x -component of forces from the cylinder, as shown in Figure 7.13b.

```
BEGIN RBE3
  REFERENCE BLOCK = BLOCK_2
  CONNECTION BLOCK = BLOCK_1
  WEIGHTS = 1 0 0 0 0 0
END RBE3
```



(a) Initial position.



(b) Final position.

Figure 7.13: RBE3 test positions.

References

- [1] K.H. Brown, J.R. Koterass, D.B. Longcope, and T.L. Warren. CavityExpansion: A library for cavity expansion algorithms, version 1.0. Technical Report SAND2003-1048, Sandia National Laboratories, Albuquerque, NM, April, 2003. [pdf](#).
- [2] J. Lysmer and R.L. Kuhlmeyer. Finite dynamic model for infinite media. In *Proceedings of the American Society of Civil Engineers*, pages 859–877, 1979.

Chapter 8

Contact

This chapter describes the input syntax for defining interactions of contact surfaces. For more information on the computational details of contact, consult References [1, 2, 3].

Contact is the interaction of bodies when they touch. Contact algorithms ensure that surfaces do not interpenetrate and that the interface behavior is computed based on specified surface physics models.

To specify contact enforcement in a model, the user must:

- Identify all surfaces to be considered for contact. Usually this is the exterior surfaces of element blocks used in the analysis.
- Specify which contact surfaces will interact with each other. This could involve tying surfaces together, allowing surfaces to slide against each other, or allowing part of a surface to slide against other parts of the same surface.

Additional algorithmic options may be included to improve the robustness and accuracy of the contact computations.

8.1 Overview of Contact Constraints

Contact between surfaces is modeled by first defining and then enforcing a set of contact constraints. A simple example of contact is shown in Figure 8.1a. Block *a* has exterior surface *a*, and block *b* has exterior surface *b*, each defined by a collection of finite element faces. For this two-dimensional drawing, the faces are straight lines between two nodes. Figure 8.1a shows the two blocks at time step *n*. Figure 8.1b shows the blocks at time step *n* + 1 when the blocks have moved and deformed under the influence of external and internal forces. Before contact is taken into account, the two blocks interpenetrate one another. This initial, potentially interpenetrated configuration is known as the initial predicted configuration. The interpenetration is removed by applying the contact algorithm.

There are three major types of contact constraints available in Sierra/SM. Node/face contact constraints are based on the concept of preventing nodes from penetrating faces on the opposing surface. Face/face constraints prevent faces on opposing surfaces from penetrating each other. Node/analytic surface constraints prevent nodes from penetrating analytically defined surfaces.

For node/face constraints, interpenetration occurs as shown in Figure 8.1b when any node on sur-

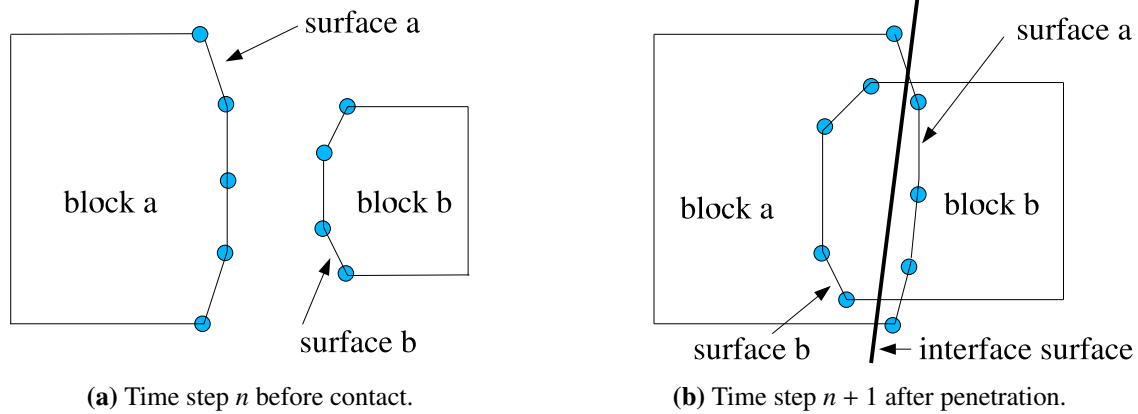


Figure 8.1: Two blocks in contact.

face a passes through a face of surface b or a node on surface b passes through a face of surface a . One option to remove interpenetration would be to force all the nodes on surface a to lie on surface b , where surface b has the configuration shown in Figure 8.1b. In this case, surface b would be a master surface and surface a would be a slave surface. Alternatively, all nodes on surface b could be forced to lie on surface a , in which case surface a is a master surface and surface b is the slave. It is generally best to let the code decide the master and slave surfaces, but the specific designations can also be input by the user.

More typically, nodes on both surfaces are moved to what can be described as an interface surface, which is shown by a thick black line in Figure 8.1b. The interface surface shown here as a straight line could be curved in reality. For each node on surface a that has penetrated a face on surface b forces based on the amount of node penetration are placed on both the node on a and the nodes associated with the face on b . Likewise, forces are computed for each node on surface b that has penetrated some face on surface a and for the nodes associated with the penetrated face on surface a . The combination of all these forces will remove the interpenetration of the two blocks and produce the final interface configuration.

The preceding two-dimensional example is analogous to much of the contact that is encountered when contact is used in an analysis. Surfaces are generated that consist of a collection of faces. Contact interactions consisting of node-face pairs are defined on the current geometry and then contact enforcement is used to remove the interpenetration of the surfaces and satisfy surface physics.

8.2 Overview of Contact Enforcement

A contact search algorithm provides the contact interactions on which contact constraints could be enforced. (An interaction in its simplest form is a node-face pairing as described in Section 8.1.) Contact enforcement must first determine which configuration dictates these interactions and which of these interactions need enforcement. Active set logic is used to determine which interactions provided by the search require interpenetration removal based on tolerances and other algorithmic issues. In addition to enforcing interpenetration constraints, the enforcement algorithm must satisfy constraint conditions defined by the friction model. In the simple example of a frictionless model, contact enforcement must ensure no penetration and a balance of forces.

The active set logic differs in the enforcement algorithms used in explicit and implicit time integration. For explicit time integration, the determination of the active set is usually made from the initial predicted position and remains unchanged for the time step. For example, in a frictionless node-face interaction, contact will enforce the constraint conditions on the pair if the node is penetrating the face in the initial predicted position. However, for large motion problems the active set can be updated within the time step by setting `NUM GEOMETRY UPDATE ITERATIONS` (See Section 8.9.7). For implicit time integration, the active set is iterated with the solve (See Section 4.5) because the motion over a time step is generally expected to be large.

Two basic algorithmic approaches exist in Sierra/SM for contact constraint enforcement. Kinematic enforcement prescribes zero interpenetration and iterates to achieve equal and opposite forces. Augmented Lagrange enforcement applies equal and opposite forces and iterates to achieve zero interpenetration. Augmented Lagrange is generally more robust, able to handle multiple conflicting constraints, and more easily incorporates complicated friction model definitions. As such, Augmented Lagrange is the default enforcement algorithm in both implicit and explicit simulations with Sierra/SM.

In explicit, the contact enforcement is linked to the search algorithm. The DASH search algorithm activates Augmented Lagrange enforcement, while ACME search activates kinematic enforcement (See Section 8.9). Both enforcement approaches use a fixed number of iterations which can be set with `MOMENTUM BALANCE ITERATIONS` (See Section 8.9.7).

In implicit, the contact enforcement algorithm is specified with the `ENFORCEMENT` command (See Section 8.9.3). For both Augmented Lagrange and Kinematic enforcement, the number of iterations is determined by convergence criteria that are specified in the `CONTROL CONTACT` section (See Section 4.5).

8.3 Contact Definition Block

The following set of commands are used to define contact. The contact definition block is placed in the `REGION` command scope. For an example of how to create a basic contact definition block, see Section 8.12. Commands which are specifically related to ACME contact are provided in Section 8.14.

```
BEGIN CONTACT DEFINITION <string>name

# Contact surface definition commands:

SKIN ALL BLOCKS = <string>ON|OFF(OFF)
  [EXCLUDE <string list> block_names]
# see Section 8.4.4

CONTACT SURFACE <string>name CONTAINS <string list>surface_names
# see Section 8.4.1

CONTACT NODE SET <string>surface_name
  CONTAINS <string>nodelist_names
# see Section 8.4.2

BEGIN CONTACT SURFACE <string>name
# ... see Section 8.4.3
END [CONTACT SURFACE <string>name]

# friction model definition commands:
# ... see Section 8.5

# interaction definition commands:

BEGIN INTERACTION DEFAULTS [<string>name]
# ... see Section 8.6.1
END [INTERACTION DEFAULTS <string>name]

BEGIN INTERACTION [<string>name]
# ... see Section 8.6.2
END [INTERACTION <string>name]

ACTIVE PERIODS = <string list>period_names
# see Section 8.7

# contact algorithm option commands:

SEARCH = <string>ACME|DASH(DASH)
# see Section 8.9.1

BEGIN DASH OPTIONS
```

```
# ... see Section 8.9.2
END
```

 **Implicit**

```
ENFORCEMENT = <string>AL|KINEMATIC(AL)
# see Section 8.9.3
```

 **Implicit**

```
CONTACT RESIDUAL NORM SCALE FACTOR = <real>scale
```

```
INITIAL OVERLAP REMOVAL = ON|OFF(OFF)
BEGIN REMOVE INITIAL OVERLAP
# ... see Section 8.9.4
END [REMOVE INITIAL OVERLAP]
OVERLAP REMOVAL ONLY = OFF|ON(OFF)
```

```
BEGIN SURFACE OPTIONS
# ... see Section 8.9.5
END [SURFACE OPTIONS]
```

```
BEGIN SEARCH OPTIONS [<string>name]
# ... see Section 8.9.6
END [SEARCH OPTIONS <string>name]
```

 **Explicit**

```
BEGIN ENFORCEMENT OPTIONS [<string>name]
# ... see Section 8.9.7
END [ENFORCEMENT OPTIONS <string>name]
```

 **Explicit**

```
COMPUTE CONTACT VARIABLES = ON|OFF|EVERY_STEP(ON)
# see Section 8.10
```

```
BEGIN DEBUG [<string>name]
# ... see Section 8.10.1
END [DEBUG <string>name]
```

```
END [CONTACT DEFINITION <string>name]
```

The command block begins with the input line:

```
BEGIN CONTACT DEFINITION <string>name
```

and is terminated with the input line:

```
END [CONTACT DEFINITION <string>name]
```

The string `name` is a name for this contact definition. The name should be unique among all the contact definitions in an analysis and is used in diagnostics printed in the log file or error messages related to the contact definition.

A typical analysis will have only one `CONTACT DEFINITION` command block. However, more than one contact definition may be used. Each `CONTACT DEFINITION` command block creates its own set of contact constraints that operate independently from one another. Use of fewer contact definition commands blocks will tend to provide more efficient processing and better conflict resolutions when different types of contact constraints come into conflict.

Contact behaves somewhat differently in explicit versus implicit analyses. Additionally there are multiple contact enforcement libraries available: ACME and Dash. The detailed descriptions of each of the commands list what options and commands are available based on what type of analysis is being done and contact library is being used. Unless otherwise noted the behavior of a command is the same regardless of the type of analysis or the contact algorithm being used.

8.4 Defining the Contact Surfaces

The user must provide a description of which surfaces will be considered for contact. There are many options available to either automatically detect and create contact surfaces or fine tune specific contact surface definitions.

Many separate contact surfaces can be defined by the analysis. Each contact surface can contain one or more of the following:

- The contact surface can include the exterior faces or “skin” of a block of finite elements. An exterior face is an outward facing finite element face that is not sandwiched between two contiguous finite elements. For more information on blocks skinning algorithms see Section 8.4.7.
- The contact surface can include a set of faces defined as a sideset on the input mesh. All faces included in the face set are considered for contact even if those faces are not exterior faces of the finite elements.
- The contact surface can include a set of nodes defined by a node set, surface, or block on the input mesh. These nodes need not be attached to finite element faces. The nodes could for example be the nodes of beam elements or particle elements.
- The contact surface can be a lofted set of faces around structural or lower dimensional elements. For example contact can define a lofted brick around a two dimensional shell element, a lofted prism around a one dimensional beam element, or lofted sphere around a zero dimensional particle element.
- The contact surface can be defined by an analytic function. More information on this capability can be found in the “Capabilities in Development” manual.

8.4.1 Contact Surface Command Line

```
CONTACT SURFACE <string>name CONTAINS <string list>surface_names
```

This command line identifies a set of sidesets and element blocks considered as a single contact surface; the string `name` is the unique name given to this contact surface. The list denoted by `surface_names` is a list of strings identifying surfaces and blocks that are to be associated with this contact surface. The surfaces can be side sets, element blocks, or any combination of the two as defined in the mesh file.

Any specified element blocks in the `surface_names` list that are solid elements (bricks, tetrahedra, wedges, etc.) are “skinned,” meaning that a surface is created from all exterior faces of that element block. See Section 8.4.7 for more information on block skinning.

For element blocks in the `surface_names` list that are lower dimensional (shells, beams, and particles), a lofted geometry is created for use with contact enforcement.



Known Issue: The ACME contact algorithm is unable to loft beam and particle elements and will ignore any beam or particle element blocks. See Section 8.9.5 for more information on lofted geometry.

The `name` you create for a surface can be referenced in command blocks that specify how that surface will interact with other contact surfaces or interact when that surface contacts itself. See Section 8.6.1.2 and Section 8.6.2.1.

The surfaces can contain heterogeneous combinations of finite element faces and lofted geometry.



Warning: A given finite element face or lofted element can be placed in at most one contact surface. See Section 8.4.5 for more information on what happens if different contact surfaces try to include the same mesh objects.

8.4.2 Contact Node Set Command Line

```
CONTACT NODE SET <string>surface_name
CONTAINS <string list>nodelist_names
```

The `CONTACT NODE SET` command line creates a contact surface named `surface_name` that contains a set of nodes. The list denoted by `nodelist_names` is a list of strings identifying node sets, surfaces, and blocks that are to be associated with the contact node set `surface_name`. All nodes present in the listed node sets, surfaces, or blocks will be added to the contact node set. The created node set contains only the mesh nodes of surfaces or blocks, not the faces, and not any lofted geometry. Thus, the contact node set can be used only in a node to face interaction.

8.4.3 Contact Surface Command Block

```
BEGIN CONTACT SURFACE <string>name
  INCLUDE ALL BLOCKS
  BLOCK = <string list>block_names
  ELEMENT = <int list>elem_numbers
  SURFACE = <string list>surface_names
  NODE SET = <string list>node_set_names
  NODE = <int list>node_numbers
  REMOVE BLOCK = <string list>block_names
  REMOVE SURFACE = <string list>surface_names
  REMOVE NODE SET = <string list>node_set_names
  SUBSET <string>subname WITH NORMAL <real> <real> <real>
END [CONTACT SURFACE <string>name]
```

The `CONTACT SURFACE` command block can be used to define a contact surface consisting of a more complex collection of finite element faces or nodes than can be defined using the `CONTACT SURFACE` command line.

A surface named `name` that is a set of faces or lofted elements can be defined by a combination of the command lines `INCLUDE ALL BLOCKS`, `BLOCK`, `SURFACE`, `ELEMENT`, `REMOVE BLOCK`, and `REMOVE SURFACE`. Blocks listed in the `block_names` list or via `INCLUDE ALL BLOCKS` will be skinned for exterior faces if the block contains solid elements (like hexahedra, tetrahedra, or wedges.) For blocks that contain shells, beams, or particles, the lofted geometry around those

shells beams and particles will be included in the contact surface. The `ELEMENT` command can be used to pull individual lofted elements into the contact surface by element id.

If a `BLOCK` command line and a `SURFACE` command line are both specified within the same `CONTACT SURFACE` command block, the contact surface created will be the union of the faces in the skin of the blocks and the faces in the specified surfaces in the mesh file.

A surface named `name` that is a set of nodes without face geometry can be defined by a combination of the command lines `NODE SET`, `NODE`, `REMOVE NODE SET`, `REMOVE SURFACE`, and `REMOVE BLOCK`. The `NODE` command can be used to pull individual nodes into the contact surface by node id.

The contact surface created by the `CONTACT SURFACE` command block must be either a set of faces and lofted geometry or a set of nodes. It cannot be a mix of the two. So for example a single `CONTACT SURFACE` block may not contain both a `NODE SET` and `SURFACE` command.

Contact surfaces may not be empty. At least one block, element, node, or surface must be defined in the input.

8.4.3.1 Surface Subsetting

```
SUBSET <string>subname WITH NORMAL <real> <real> <real>
```

A contact surface can optionally be subdivided into multiple sub-surfaces based on the surface normal. This command will create a new surface that contains the faces mostly aligned with the specified normal direction. Faces for which the dot product of the unitized face normal vector and the unitized specified normal is greater than 0.45 are placed in the new surface. All other faces are left in the original surface. For example, the following command:

```
BEGIN CONTACT SURFACE surf1
  BLOCK = block_1
  SUBSET xp WITH NORMAL 1 0 0
  SUBSET zm WITH NORMAL 0 0 -1
END
```

will create three contact surfaces. Surface `surf1_xp` will contain the faces most closely aligned with the positive x-axis. Surface `surf1_zm` will contain the faces mostly aligned with the negative z-axis. Surface `surf1` will contain all remaining faces in the skin of `block_1`.

8.4.3.2 Known Issues

The `BEGIN CONTACT SURFACE` block may sometimes resolve contact faces slightly differently than the basic `SKIN ALL BLOCKS` or `CONTACT SURFACE` command. One potential distinction is faces interior to the model. When using global block skinning a face sandwiched between two element blocks is not considered for contact. When faces are constructed with a more complex combination of individual block skins plus or minus side sets, those in-between block faces might be considered as contact faces. Generally the Dash contact enforcement algorithm will ultimately ignore any spurious internal faces if they exist, but these internal faces may cause fatal errors for the ACME contact enforcement algorithm. This consistency issue may be resolved in the future if

as an unambiguous definition of how the skinning algorithm should behave in all addition/removal cases is developed.

8.4.4 Skin All Blocks Command

```
SKIN ALL BLOCKS = <string>ON|OFF(OFF)
                  [EXCLUDE <string list>block_names]
```

The `SKIN ALL BLOCKS` command provides a convenient way to create a number of contact surfaces covering the exterior skin or lofted geometry of all or most of the element blocks in the analysis. This command may create many surfaces, where the name of each surface is the same as the element block it was generated from. The optional `EXCLUDE` keyword can be used to skin all element blocks except the listed blocks. For more information on how block skinning works see [Section 8.4.7](#)

The `SKIN ALL BLOCKS` is useful for large models in which individual specifying each contact surfaces would be unwieldy. The behavior of skin all blocks is best described by example. If a finite element analysis contains five element blocks:

- `block_1` (eight-noded hexahedral elements)
- `block_3` (tetrahedral elements)
- `block_4` (shell elements)
- `block_5` (beam elements)

The command:

```
SKIN ALL BLOCKS = ON EXCLUDE block_2
```

Would be equivalent to defining several independent surfaces with:

```
CONTACT SURFACE block_1 CONTAINS block_1
CONTACT SURFACE block_3 CONTAINS block_3
CONTACT SURFACE block_4 CONTAINS block_4
CONTACT SURFACE block_5 CONTAINS block_5
```

The generated contact surfaces would be named `block_1`, `block_3`, `block_4`, and `block_5`. The contact surfaces named `block_1` and `block_3` would contain the exterior faces of the solid element blocks `block_1` and `block_3`, respectively. The contact surfaces named `block_4` and `block_5` would contain lofted geometry around the shell and beam elements in `block_4` and `block_5`, respectively.

The `SKIN ALL BLOCKS` command can be combined with any of the other surface definition methods available in the `CONTACT DEFINITION` command block.

8.4.5 Faces and Nodes in Multiple Contact Surfaces

The contact enforcement algorithms in Sierra/SM only allow for a face, node, or lofted object to be associated with a single contact surface. If such a contact entity were allowed to belong to more

than one contact surface, ambiguities would arise in determining the interaction properties for that entity.

A situation where an entity could potentially be placed in more than one contact surface is when overlapping sidesets are used in contact. This could also occur if a combination of skinned block surfaces, side sets, and node sets on the mesh file are used to define a contact surface. The side sets can potentially include the same finite element faces as the block skin.

If a given finite element face or node is potentially part of more than one contact surface, then that face or node is included in the first contact surface to which it belongs that is listed in the `CONTACT DEFINITION` block. If the same face or node also potentially belongs to another contact surface listed later in the `CONTACT DEFINITION` block, it will not be included again in this later contact surface. A warning message is generated in the log file for any potentially ambiguous cases.

Use of overlapping contact surfaces can be used to implement more advanced behavior such as surfaces tied together at local patches.

The behavior described above is best illustrated by the following examples.

Example 1: Consider the model setup shown in Figure 8.2a. In the contact definition block, the contact surfaces are defined by the following input commands:

```
contact surface surfB contains surface_2
contact surface surfA contains block_1
```

This results in a set of surfaces as shown in Figure 8.2b. The contact surface “surfB” is listed first in the input deck. The faces on side set 2 are placed into surfB, as surfA is listed second in the input deck. The remaining faces on the skin are placed in surfA. This means that the faces that are in both surface_2 and in block_1 are only placed in surfB. The sub-setting mechanism shown in this example may be used to specify a different friction model, such as tied contact, on a small piece of a larger part.

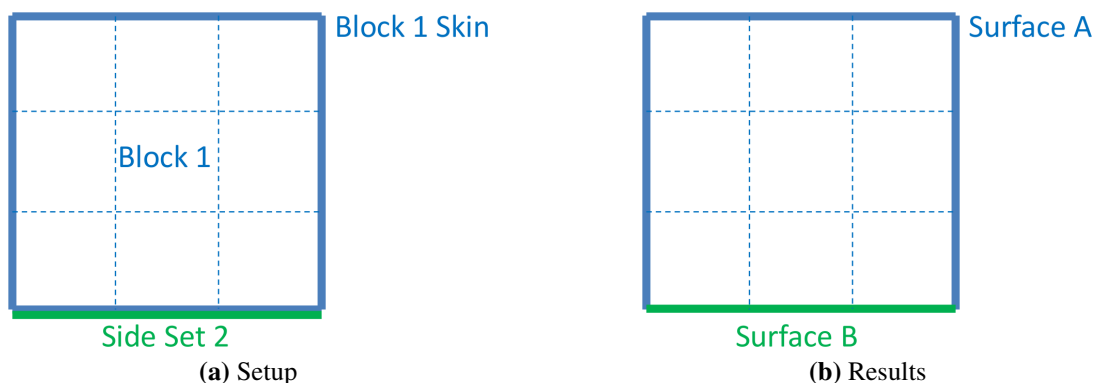


Figure 8.2: Faces in Multiple Contact Surfaces: Example 1

Example 2: A more complex example is shown in Figure 8.3a. Consider the following two contact surface definitions:

Contact surface definition 1:

```

contact surface surfD contains surface_4
contact surface surfB contains surface_2
contact surface surfA contains block_1
contact surface surfC contains block_2

```

Contact surface definition 2:

```

contact surface surfB contains surface_2
contact surface surfA contains block_1
contact surface surfC contains block_2
contact surface surfD contains surface_4

```

The first contact definition results in the organization of surfaces, as shown in Figure 8.3b. The second contact definition results in a different organization of the contact surfaces, as shown in Figure 8.3c. Unlike most commands, the order in which the contact surface definition commands are placed in the input deck may change the analysis behavior. One final note, contact surface definition two above would also be equivalent to the following:

```

contact surface surfB contains surface_2
skin all blocks = on
contact surface surfD contains surface_4

```

If the skin all blocks command is present, that command is equivalent to specifying contact on all blocks individually.

8.4.6 Element Death, Remeshing, and Surface Updates

When elements are killed in an analysis, contact surfaces may need to be updated to account for the removal of faces attached to killed elements or the addition of faces exposed by element death. The updates happen automatically when elements in a skinned block are removed due to element death (See Section 6.5). The update of these surfaces requires a full re-initialization of contact. Thus, surface-physics models that involve state data may lose some information when the new contact surfaces are created. If contact surfaces are defined by side sets rather than by block skinning, element death will cause the facets attached to killed elements to be removed from the contact surface, but no new facets will be added.

8.4.7 Block Skinning and Surface Definition Examples

A surface is defined as a collection of finite element faces, nodes, and lofted geometry. For a continuum element, any face that is not shared with another element can be considered for contact. A shell element has both a top face and a bottom face. Top, bottom, and side surfaces of shells are automatically created by the contact algorithm and may be lofted by a user-specified thickness. Contact is enforced on shells by computing the contact forces on the top and bottom surfaces of the shells and then moving the resulting forces back to the original shell nodes.

The easiest method to create contact surfaces is by skinning blocks. Block skinning identifies the exterior surfaces of a block and puts them in a contact surface. The following examples demonstrate how block skinning works in various circumstances.

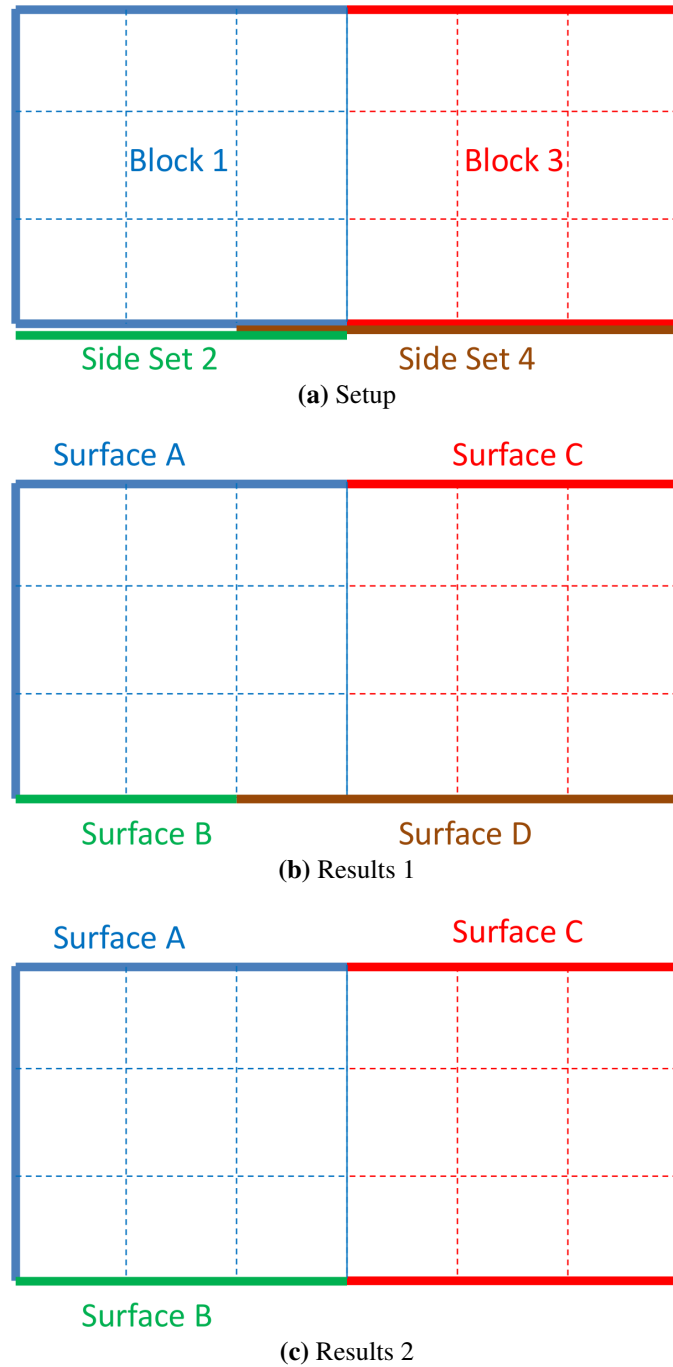


Figure 8.3: Faces in Multiple Contact Surfaces: Example 2

8.4.7.1 Simple Block Skinning

See Figure 8.4. This first example is a single block being skinned. This skinning could occur by one of the input lines

```
CONTACT SURFACE block_1 CONTAINS block_1
```

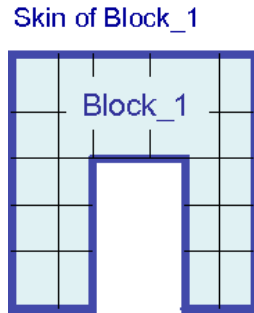


Figure 8.4: Simple block skinning example

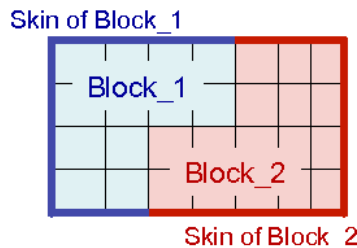


Figure 8.5: Block skinning with multiple blocks example

or if the model contains only `block_1`

```
SKIN ALL BLOCKS = ON
```

For either syntax all the exterior facets of `block_1` are automatically identified and are placed into a contact surface which is also named `block_1`.

8.4.7.2 Block Skinning With Multiple Element Blocks

Figure 8.5 shows an example of two blocks that are equivalenced on their interface. An equivalenced interface means that the two blocks share nodes at the interface so that there are no exposed contact faces between `block_1` and `block_2`. The syntax to create the contact surfaces shown in Figure 8.5 is:

```
CONTACT SURFACE block_1 CONTAINS block_1
CONTACT SURFACE block_2 CONTAINS block_2
```

Alternatively, if the model contains only `block_1` and `block_2`, this could be accomplished by requesting that all blocks be skinned:

```
SKIN ALL BLOCKS = ON
```

For either syntax, all the exterior facets that are connected to elements that are part of `block_1` are placed in contact surface `block_1`. All the exterior facets that are connected to elements of `block_2` are placed in contact surface `block_2`. Any interior facet (one that is sandwiched between two elements) is not placed in any contact surface.

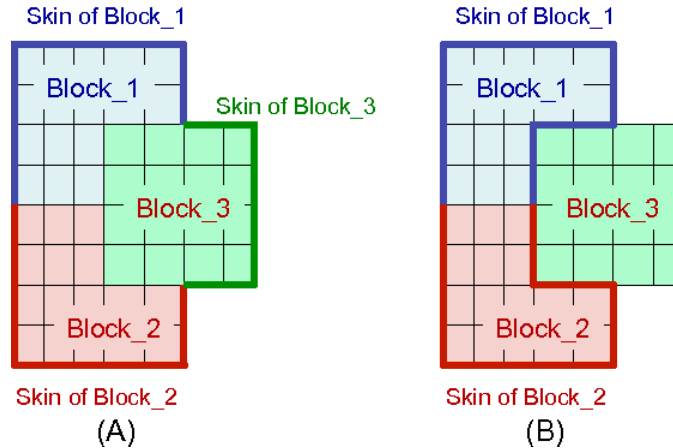


Figure 8.6: Block skinning advanced example

8.4.7.3 More Complex Block Skinning Cases

Block skinning considers only those elements actually present in the contact definition when deciding if a face is exposed. The surfaces shown by heavy lines in Figure 8.6(A) will result from the following syntax:

```
CONTACT SURFACE block_1 CONTAINS block_1
CONTACT SURFACE block_2 CONTAINS block_2
CONTACT SURFACE block_3 CONTAINS block_3
```

All the exterior facets of the model have been placed in the contact surfaces associated with elements of the named blocks. Any facet sandwiched between two contact surface blocks is not considered to be on the exterior, and is thus not placed in any contact surface. The surfaces shown in Figure 8.6(B) will result from the following syntax:

```
CONTACT SURFACE block_1 CONTAINS block_1
CONTACT SURFACE block_2 CONTAINS block_2
```

or

```
SKIN ALL BLOCKS = ON EXCLUDE block_3
```

In this case, `block_3` was not included in the contact definition. Thus the faces on the `block_1`, `block_3` and `block_2` and `block_3` boundaries will be considered exterior for the purposes of contact. Since `block_3` is not part of the contact definition none of the remaining faces of `block_3` will be part of any contact surface.

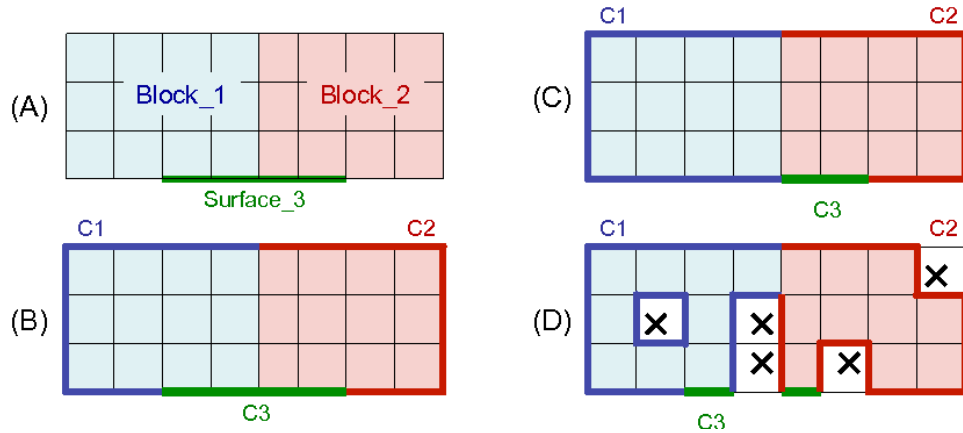


Figure 8.7: Contact surface definition examples

8.4.7.4 Combining Block Skinning, Side Sets, and Element Death

Figure 8.7(A) shows a set of element blocks and surface sidesets as defined in a mesh file. Blocks `block_1` and `block_2` are equivalenced sharing nodes on their boundary. The surface `surface_3` is a sideset defined on part of the exterior of `block_1` and `block_2`. The contact surfaces shown in Figure 8.7(B) will be generated by the following syntax:

```
CONTACT SURFACE c3 CONTAINS surface_3
CONTACT SURFACE c1 CONTAINS block_1
CONTACT SURFACE c2 CONTAINS block_2
```

As described in Section 8.4.5, the first surface defined in the input file that includes a facet owns that facet. If any subsequently defined contact surfaces attempt to include a facet that already belongs to a contact surface, it will be left in the original contact surface that it belongs to, and will not be made part of the surface that is defined later in the input file. In other words, a given mesh facet can belong in at most one contact surface, and when a conflict arises, the facet will belong to the first contact surface in the input file that included it. Thus, `c3` contains the facets associated with `surface_3`. Contact surfaces `c1` and `c2` do not contain any of the facets of side set `surface_3`.

The surfaces in Figure 8.7(C) are be generated by the following syntax:

```
CONTACT SURFACE c1 CONTAINS block_1
CONTACT SURFACE c3 CONTAINS surface_3
CONTACT SURFACE c2 CONTAINS block_2
```

Since contact surface `c1` is listed before contact surface `c3` in the input file, `c1`, the skin of `block_1`, contains the contact facets that could have been part of `c1` or `c3`. Since contact surface `c3` is listed before contact surface `c2` in the input file, contact surface `c3` includes the remaining portions of `surface_3`, while `c2`, the skin of `block_2`, contains only the remaining exterior facets of `block_2`.

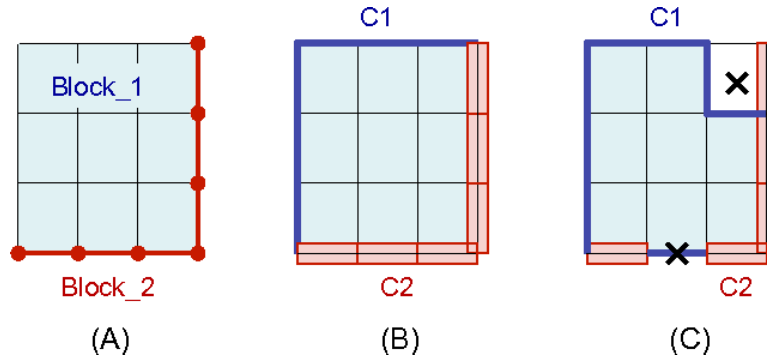


Figure 8.8: Hexahedral and shell contact surface definitions

When element death occurs, new contact facets may be exposed and old contact facets may be deleted. After element death, the ownership of contact facets in the updated contact surfaces will be evaluated as if the dead elements never existed. Figure 8.7(D) shows the effect of element death on the contact surfaces. For this case the contact surfaces are defined by the same syntax as the case shown in Figure 8.7(B):

```
CONTACT SURFACE c3 CONTAINS surface_3
CONTACT SURFACE c1 CONTAINS block_1
CONTACT SURFACE c2 CONTAINS block_2
```

Elements shown in the figure with an X are dead elements. Any newly exposed facets of `block_1` and `block_2` are added to the relevant contact surfaces. If all the elements attached to contact facet are deleted, the contact facet itself is also deleted. If a contact surface is defined via a side set, that surface can only lose faces. In other words, a contact surface defined by a contact side set cannot gain any faces that were not part of the original side set during the process of element death.

8.4.7.5 Equivalenced Solid and Shell Meshes

A mesh that includes both solid elements such as hexahedral and shell elements obeys some special rules when defining contact surfaces via block skinning. Figure 8.8(A) shows `block_1`, (solid hexahedral elements) and `block_2` (quadrilateral shell elements). The nodes of the shell element block are equivalenced with the nodes of the hexahedral element block, or in other words, the two blocks share nodes on their interface. Figure 8.8(B) shows the contact surfaces that would be generated using the following commands:

```
CONTACT SURFACE c1 CONTAINS block_1
CONTACT SURFACE c2 CONTAINS block_2
```

No matter what the order of the contact surfaces in the input file is, the shell block will always be considered “outside” the solid element block for this case. On portions of `block_1` without any cladding shells, the standard block skinning rules apply. On the interface between `block_1` and

`block_2` the quad faces on the `block_1` hexahedral elements are considered sandwiched between the `block_1` hexahedral elements and the `block_2` shell elements. Thus, `c1` contains none of the interface facets. Contact surface `c2` contains the lofted shell geometry associated with the shell elements of `block_2`.

In Figure 8.8(C), some elements have been removed due to element death. As in the example in Section 8.4.7.4, when elements are killed via element death the contact surfaces are reevaluated as if those elements were never there. When a cladding shell is removed the exterior face of `block_1` is exposed and joins contact surface `c1`. The death of one of the lofted shell elements removes all faces associated with that element's lofted geometry.

8.5 Friction Models

Friction models are used to describe the physics of interactions that occur between contact surfaces. The user then associates these friction models to pairs of surfaces in the interaction-definition blocks (see Section 8.6.1 and Section 8.6.2). During the search phase of contact, node-face or face-face interactions are identified, and the designated friction model is used to determine how the resulting contact forces are resolved between these surface pairs.

Not all friction models are available with all analysis types and all contact libraries. If a model is not available for certain analysis types, that fact is noted in the documentation of those models below.

In addition to the friction models, any of the cohesive zone models available for use with interface elements (see Section 5.3) can be used as interface models within explicit dynamic analyses. See Section 8.6.2.5 for more details on the usage of those models within contact.

All friction models are command blocks, although some of the models do not have any command lines inside the command block. The commands for defining the available friction models are described next. Friction models are associated with specific pairings of contact surfaces through the interaction-definition blocks in Section 8.6.1 and Section 8.6.2.

By default, interactions between contact surfaces with no friction model assigned are treated as frictionless.

8.5.1 Frictionless Model

```
BEGIN FRICTIONLESS MODEL <string>name  
END [FRICTIONLESS MODEL <string>name]
```

The `FRICTIONLESS MODEL` command block defines frictionless contact between surfaces. In frictionless contact, contact forces are computed normal to the contact surfaces to prevent penetration, but no forces are computed tangential to the contact surfaces. The string `name` is a user-selected name for this friction model that is used when identifying this model in the interaction definitions. No command lines are needed inside the command block. A default named frictionless model named `frictionless` can be used without defining this command block.

8.5.2 Constant Friction Model

```
BEGIN CONSTANT FRICTION MODEL <string>name  
  FRICTION COEFFICIENT = <real>coeff  
END [CONSTANT FRICTION MODEL <string>name]
```

The `CONSTANT FRICTION MODEL` command block defines a constant coulomb friction coefficient between two surfaces as they slide past each other in contact. No resistance is provided to keep the surfaces together if they start to separate. The string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions, and `coeff` is the constant coulomb friction coefficient. There is no default value for the friction coefficient.

8.5.3 Velocity Dependent Coulomb Friction Model



Explicit Only



Explicit Only

```

BEGIN VELOCITY DEPENDENT COULOMB MODEL <string>name
  COULOMB COEFFICIENT = <string>coeff_func
END [VELOCITY DEPENDENT COULOMB MODEL <string>name]

```

The `VELOCITY DEPENDENT COULOMB MODEL` command block defines a Coulomb friction law between two surfaces as they slide past each other in contact. The value of the Coulomb coefficient at any given time is a function of the relative velocity at the interface. No resistance is provided to keep the surfaces together if they start to separate. The string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions. The `COULOMB COEFFICIENT` command provides `coeff_func`, the name of a previously defined XY function, where X is the velocity magnitude and Y is the coulomb coefficient. The velocity dependent coulomb model can be used to define stick-slip friction behavior where the friction coefficient is the stick value when velocity is nominally small and the slip value when velocity is larger than some nominal value.



8.5.4 Variable Coefficient Coulomb Model

```

BEGIN VARIABLE COEFFICIENT COULOMB MODEL <string>name
  FUNCTION ARGUMENT: <name> = MASTER NODAL <var_name>
  FUNCTION ARGUMENT: <name> = SLAVE NODAL <var_name>
  COEFFICIENT FUNCTION      = <string>coeff_func
END VARIABLE COEFFICIENT COULOMB MODEL <string>name]

```

The `VARIABLE COEFFICIENT COULOMB MODEL` command block can be used to define a Coulomb friction law where the friction coefficient is dependent on some arbitrary nodal variable. For example, a different coefficient based on spatial coordinates or from some surface dependent property. See section [2.1.6](#) for more information on generalized functions. An example of using this friction model to compute a friction model that depends on an oxidation state nodal surface variable follows.

```

#
# Friction coefficient based on oxide state. Assume
# oxide thickness < 1.0e-6 means "clean" surface
# and thickness > 1.0e-6 is a rough oxidized surface.
# Friction coefficient is then subject to following rules:
#
# oxide against oxide: u = 0.4
# oxide against clean: u = 0.2
# clean against clean: u = 0.1
#
begin function oxide_fric_coeff
  type is analytic
  expression variable: ox1
  expression variable: ox2
  evaluate expression = "      #
    (ox1 < 1.0e-6) ? (      #

```

```

        (ox2 < 1.0e-6) ? (      #
            0.1                #
        ) : (                  #
            0.2                #
        )                      #
    ) : (                      #
        (ox2 < 1.0e-6) ? (      #
            0.2                #
        ) : (                  #
            0.4                #
        )                      #
    )                          #
"
end

begin variable coefficient coulomb model oxide_fric
    coefficient function = oxide_fric_coeff
    Function Argument: ox1 = master nodal oxide_thick
    Function Argument: ox2 = slave  nodal oxide_thick
end

```

8.5.5 Tied Model

```

BEGIN TIED MODEL <string>name
END [TIED MODEL <string>name]

```

The `TIED` friction model attaches together surfaces that are close in the initial model geometry. Tied faces will stay attached throughout the analysis. The string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions. No command lines are needed inside the command block. A default tied model named `tied` can be used without defining this command block.

8.5.6 Inactive Friction Model

```

BEGIN INACTIVE FRICTION MODEL <string>name
END [INACTIVE FRICTION MODEL <string>name]

```

The `INACTIVE FRICTION MODEL` command block specifies that the interacting surfaces produce no force and contact is effectively ignored on the interaction pair. The inactive friction model can be used to selectively turn off interaction pairs. One common use is integrating the inactive friction model to a `TIME VARIANT` friction model to turn contact interactions off selectively by time period.

8.5.7 Dynamic Tied Model

```

BEGIN DYNAMIC TIED MODEL <string>name
END [DYNAMIC TIED MODEL <string>name]

```



The `DYNAMIC TIED MODEL` command block defines a contact interaction that constrains opposing faces to stay in the same relative location to each other throughout the analysis. No command lines are needed within this command block. A default dynamic tied model named `dynamic_tied` can be used without defining this command block. The dynamic tied model is similar to the tied model except that the dynamic tied model enforces zero relative velocity at the interface while the tied model enforces zero relative displacement. In the limit, both of these models are identical. If the analysis involves substantial topology changes, such as element death or remeshing, the dynamic tied model may be more stable, but slightly less accurate.

8.5.8 Glued Model

```
BEGIN GLUED MODEL <string>name
END [GLUED MODEL <string>name]
```

Note: This model is available for explicit and implicit analyses, but for implicit analyses, it can only be used with augmented Lagrange enforcement.

The `GLUED` model defines a contact interaction that allows the interacting faces to move independently until they come into contact, but once they come into contact they become permanently attached together. Once attached the faces can have no relative normal or tangential motion for the rest of the analysis. The string `name` is a user-specified name for this friction model that is used to reference this model in the interaction definitions. No command lines are needed inside the command block. A default named glued model named `glued` can be used without defining this command block.

The glued model is enforced via a zero velocity constraint.

8.5.9 Cohesive Zone Model

```
BEGIN COHESIVE ZONE MODEL <string>name
  TRACTION DISPLACEMENT FUNCTION = <string>func_name
  TRACTION DISPLACEMENT SCALE FACTOR = <real>scale_factor(1.0)
  CRITICAL NORMAL GAP = <real>crit_norm_gap
  CRITICAL TANGENTIAL GAP = <real>crit_tangential_gap
END [COHESIVE ZONE MODEL <string>name]
```

The `COHESIVE ZONE MODEL` command block defines a friction model that prevents penetration when contact surfaces are touching, and provides an additional force when the distance between the node and face in an interaction increases. This force is determined by a user-specified function. Once the distance exceeds a user-specified value in the normal direction or the tangential direction, the force is no longer applied. This model can be used to simulate the energy required to separate two surfaces that are initially touching.

In the above command block, the string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions. The traction-separation relationship is given by the `TRACTION DISPLACEMENT FUNCTION` command line, where the string `func_name` is the name of a function defined in a `FUNCTION` command block in the SIERRA scope. The y-values of this function can be scaled by the real value `scale_factor`

in the `TRACTION DISPLACEMENT SCALE FACTOR` command line; the default for this factor is 1.0. In the `CRITICAL NORMAL GAP` command line, the real value `crit_norm_gap` specifies the normal distance between the node and face past which the cohesive zone no longer provides a force. Similarly, in the `CRITICAL TANGENTIAL GAP` command line, the real value `crit_tangential_gap` specifies the tangential distance between the node and face past which the cohesive zone no longer provides a force.

The function referenced by the `TRACTION DISPLACEMENT FUNCTION` command line must be an XY function (see Section 2.1.5). The X-values correspond to ϵ , defined as:

$$\epsilon = \frac{\text{current displacement}}{\text{critical gap length}} \quad (8.1)$$

ϵ is a non-dimensional quantity which should start at zero corresponding to zero traction, and end at 1.0 corresponding to the critical gap length. The Y-values are the ϵ -dependent tractions between faces sharing a cohesive zone. All traction and separation quantities should be positive.

The following example is the problem shown in Figure 8.9: 2 bricks being pulled apart that are initially connected by a cohesive zone.

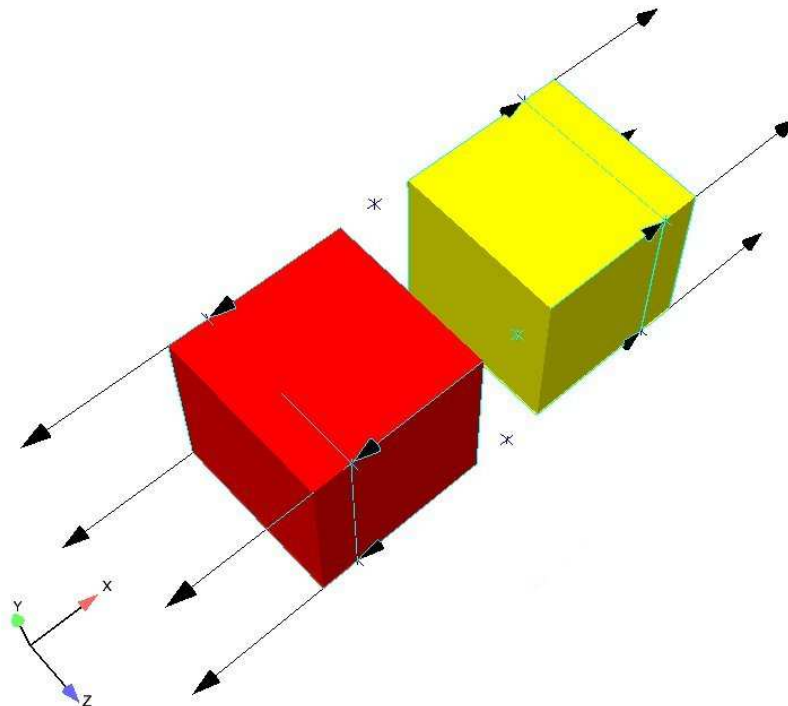


Figure 8.9: Bricks sharing a cohesive zone and being pulled apart.

The function describing the cohesive zone is given by:

```
begin function spring_restore
  type = piecewise linear
```

```

begin values
  # separation(epsilon) traction
  0.0    0.0
  0.1    1.0
  0.3    9.0
  0.4   16.0
  0.5   25.0
  0.9   81.0
  1.0  100.0
end values
end function spring_restore

```

The cohesive zone model is given by:

```

begin cohesive zone model cohesive_zone
  critical normal gap = 0.05
  critical tangential gap = 0.05
  traction displacement function = spring_restore
  traction displacement scale factor = 2.5E+02
end cohesive zone model cohesive_zone

```

A plot of the x-displacement for node 8 (one of the nodes on the cohesive zone) over time is given in Figure 8.10. The corresponding cohesive force in the x-direction over time for node 8 is shown in Figure 8.11 .

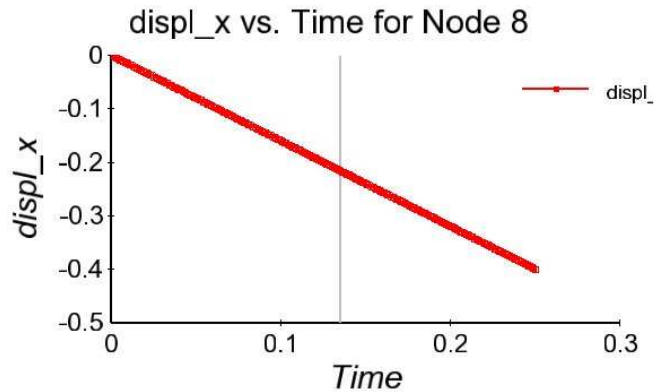


Figure 8.10: Displacement X vs. time for node 8

As expected, when the displacement exceeds the critical normal gap of .05, which happens at .016 seconds, the cohesive force goes to zero. The cohesive zone force peaks at 6250, which is equal to:

$$\frac{\text{force at critical gap} \times \text{traction displacement scale factor}}{\text{number of nodes on face}} = \frac{100 \times 250}{4}. \quad (8.2)$$

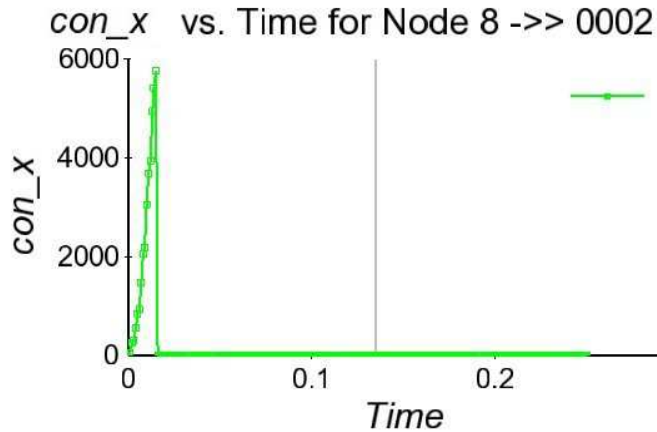


Figure 8.11: Cohesive force in x vs. time for node 8

8.5.10 Hybrid Model

```
BEGIN HYBRID MODEL <string>name
  INITIALLY CLOSE = <string>close_name
  INITIALLY FAR   = <string>far_name
END [HYBRID MODEL <string>name]
```

The `HYBRID MODEL` command block defines a single friction model that is a composite of two other models: one for faces that are initially close together, and one for faces that are initially far apart. The friction law specified in the `INITIALLY CLOSE` command is used for faces that are close together in the initial mesh geometry. The friction law specified in the `INITIALLY FAR` command is used for faces that are far apart in the initial mesh geometry.

A common usage of the hybrid model is to tie together parts of two surfaces that are close together, while allowing the remaining parts of those surfaces to have a sliding contact interaction. If the tied model were used in that situation, the faces that are initially close would be tied together, but no contact would be enforced on the remaining faces on those surfaces, so they could pass through one another. Another potential usage of the hybrid model would be to define a high friction coefficient for faces that are initially close, and then use a lower friction coefficient once the faces start to slide away from the initial configuration.

A hybrid model is automatically created when an interaction-specific and a default friction model are both specified and the interaction-specific friction law is only applicable when the surfaces are initially close together. For example, the hybrid model contact interaction definition:

```
BEGIN HYBRID MODEL hmod
  INITIALLY CLOSE = tied
  INITIALLY FAR   = frictionless
END

BEGIN INTERACTION
  SURFACES = block_1 block_2
```

```

friction model = hmod
END

```

Is equivalent to:

```

BEGIN INTERACTION DEFAULTS
  FRICTION MODEL = frictionless
END
BEGIN INTERACTION
  SURFACES = block_1 block_2
  friction model = tied
END

```

The hybrid model definition above was used for the example model shown in Figure 8.12.

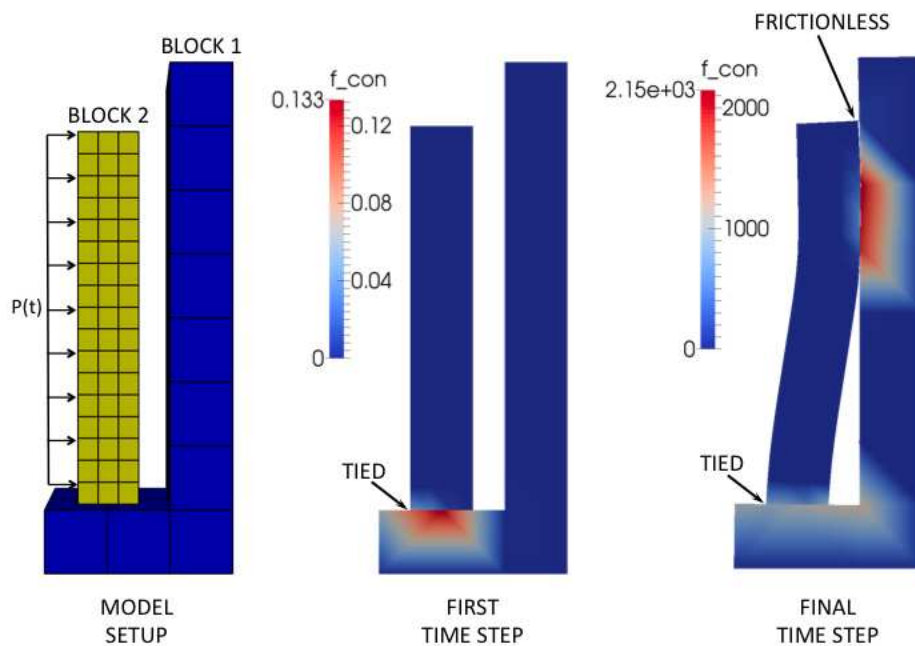


Figure 8.12: Example usage of the hybrid friction model. The contour plots show the magnitude of the contact force.

8.5.11 Time Variant Model

```

BEGIN TIME VARIANT MODEL <string>name
  MODEL = <string>model DURING PERIODS <string list>time_periods
END [TIME VARIANT MODEL]

```

Note: This feature is only available for Dash contact with explicit dynamics and Dash augmented Lagrange contact for implicit analyses.

The `TIME VARIANT MODEL` command block does not directly define a friction model, but provides a mechanism to switch between separately defined friction models between time periods. The only command line in the block is the `MODEL` command, which is used to reference a separately-defined friction model `model` and provide a list of time periods during which that model is active. Multiple `MODEL` commands may be used to activate different friction models throughout the analysis. See Section 2.6 for more information on controlling functionality by time period.

8.5.12 Nodal Field Dependent Model

```
BEGIN NODAL FIELD DEPENDENT FRICTION MODEL <string>name
  INITIAL CONTACT TRANSITION VALUE = <real>transition_value
  CONTACT TRANSITION REDUCTION METHOD =
    <string>MIN|MAX|INTERPOLATE (MAX)
END [NODAL FIELD DEPENDENT FRICTION MODEL]
```

Note: This feature is only available for Dash contact with explicit dynamics and Dash augmented Lagrange contact for implicit analyses.

The `NODAL FIELD DEPENDENT FRICTION MODEL` command block allows the opportunity to vary the friction coefficient and switch between Coulomb and glued friction conditions based on the value of a specific nodal variable, named `contact_transition`. For every interaction pair for which this friction model is applied, the `contact_transition` field is evaluated on the slave nodes to determine the type of contact: a non-negative `contact_transition` value results in Coulomb contact and a negative value results in glued contact. For Coulomb contact, the coefficient of friction is set equal to the `contact_transition` value.

The `contact_transition` field is controlled by the user. The input command `INITIAL CONTACT TRANSITION VALUE` sets an initial value for all nodes in the model. The field can then be varied spatially and temporally by a `USER OUTPUT` command block (Section 9.3.2), where `contact_transition` is the variable name. This allows the user to define `contact_transition` as a function of other field variables. For example, the following input commands make the contact conditions dependent on the temperature of element block `block_1`:

```
BEGIN FUNCTION my_temperature
  TYPE IS PIECEWISE ANALYTIC
  BEGIN EXPRESSIONS
    # time    temperature
    0.0       "600.0"
    0.00006   "600.0+1400.0*(x-0.00006)/0.00394"
    0.004     "2000.0-1600.0*(x-0.004)/0.0005"
    0.0045    "400.0"
  END
END FUNCTION my_temperature

BEGIN FUNCTION temp_dependent_fric
  TYPE IS ANALYTIC
  EXPRESSION VARIABLE: q = GLOBAL temperature_max
```

```

    EVALUATE EXPRESSION IS "q > 500.0 ? (2000.0-q)/4000.0 : -1.0"
END FUNCTION temp_dependent_fric

BEGIN PRESCRIBED TEMPERATURE
    BLOCK = block_1
    FUNCTION = my_temperature
END PRESCRIBED TEMPERATURE

BEGIN USER OUTPUT
    BLOCK = block_1
    COMPUTE GLOBAL temperature_max AS MAX OF NODAL temperature
END USER OUTPUT

BEGIN USER OUTPUT
    BLOCK = block_1 block_2
    COMPUTE NODAL contact_transition AS FUNCTION temp_dependent_fric
END USER OUTPUT

BEGIN CONTACT DEFINITION slide_and_glued
    SEARCH = DASH
    SKIN ALL BLOCKS = ON

    BEGIN INTERACTION
        SURFACES = block_1 block_2
        FRICTION MODEL = temperature_dependent
    END INTERACTION

    BEGIN NODAL FIELD DEPENDENT FRICTION MODEL temperature_dependent
        INITIAL CONTACT TRANSITION VALUE = 0.35
        CONTACT TRANSITION REDUCTION METHOD = MAX
    END NODAL FIELD DEPENDENT FRICTION MODEL temperature_dependent
END CONTACT DEFINITION slide_and_glued

```

In this example, the friction conditions are Coulomb at above 500.0 degrees and glued at 500.0 degrees or less. Above 500.0 degrees, the friction coefficient varies linearly with temperature. The temperature of `block_1` starts out above 500.0 degrees and varies significantly causing a variation in the friction coefficient, until a time just prior to 0.0045 when the friction conditions become glued. The temperature of `block_1` is stored as the global variable `temperature_max` so that both blocks involved in the interaction, `block_1` and `block_2`, will have the correct `contact_transition` value, which is computed by calling the `temperature_dependent_fric` function in a `USER OUTPUT` command block. By default, the Dash algorithms internally determine the master and slave surfaces of a contact interaction, so it is necessary to set the `contact_transition` field to the desired values on both surfaces involved in the interaction.

The input command `CONTACT TRANSITION REDUCTION METHOD` defines how the nodal `contact_transition` field is reduced to a single scalar at the point of interaction inside a face. The `MIN` and `MAX` options take the minimum and maximum, respectively, of the face's nodal values.

The `INTERPOLATE` option interpolates the nodal values to the point of interaction.

8.6 Definition of Interactions

Each contact surface can interact with itself and other contact surfaces subject to the conditions specified in a contact interaction. A contact interaction must specify which surfaces should be considered for interaction and parameters governing the interaction.

Defining contact in complex models can be greatly simplified by using “general” contact, in which a single command is used to indicate possible contact between one contact surface and every other contact surface in the model. This command used with a default interactions block to define parameters for the interaction behavior can define contact for an entire model in only a few lines of input.

It is recommended that default interactions blocks be used as much as possible to keep the definition of individual interactions to a minimum. However, if needed, contact interactions can also be defined individually for specific sets of contact surfaces. These can be used to override parameters set in the default command block or can be used without specifying any default parameters. The latter may be more efficient if there are relatively few surfaces in a model and those surfaces interact with a small number of other surfaces, but for complex models with many potential contact surfaces, individually defining all possible contact interactions can be a tedious, error-prone process.

Contact can also be enforced between a surface and itself. This is known as “self” contact, and is used to prevent a part of a surface from penetrating another part of that surface when it folds in on itself. As a general rule, enforcing self contact is more expensive than enforcing contact between different surfaces.

Contact surfaces can be based either on sets of nodes or faces. Node-based surfaces can only interact with face-based surfaces. Face-based surfaces can interact with node or face based surfaces.

The following sections document the commands used to define contact interactions and the properties of those interactions.

8.6.1 Interaction Defaults Block

```
BEGIN INTERACTION DEFAULTS
  GENERAL CONTACT = <string>ON|OFF(OFF)
  SELF CONTACT = <string>ON|OFF(OFF)
  SURFACES = <string list>surface_names
  FRICTION MODEL = <string>friction_model_name(FRICTIONLESS)
  INTERACTION BEHAVIOR = <string>SLIDING|NO_INTERACTION(SLIDING)
  CONSTRAINT FORMULATION = <string>NODE_FACE|FACE_FACE
  AL PENALTY = <real>normalPenalty(1.0) [<real>tangentialPenalty]
END [INTERACTION DEFAULTS]
```



The `INTERACTION DEFAULTS` command block enables contact enforcement either on all contact surfaces or on a subset of the contact surfaces. It is also used to set default parameters for contact interactions. These defaults can be overridden for specific interactions by defining them in `INTERACTION` blocks. Note that contact enforcement will not occur in a model unless some combination of the `INTERACTION DEFAULTS` command block and `INTERACTION` command blocks

(see Section 8.6.2) is used to set up surface interactions.

The values specified by the command lines in the `INTERACTION DEFAULTS` command block are applied by default to all interaction contact surfaces unless overridden by a specific interaction definition. Note that if a friction model is specified for a specific interaction, but that friction model is relevant only for some of the faces in the surfaces involved in the interaction, the friction model specified in the interaction defaults will be used for the remaining faces. For example, if a tied model is specified for an interaction, only the faces that are initially close will be tied. The remaining faces will be enforced using the default friction model. See Section 8.5.10 for more information.

8.6.1.1 General Contact and Self-Contact

```
GENERAL CONTACT = <string>ON|OFF (OFF)
SELF CONTACT = <string>ON|OFF (OFF)
```

Setting `GENERAL CONTACT` to `ON` specifies that default values set in the command lines of this `INTERACTION DEFAULTS` command block apply all surfaces defined in the enclosing `CONTACT DEFINITION` block or to a subset of those surfaces specified in a `CONTACT SURFACES` line. The default value is `OFF`. The `GENERAL CONTACT` command does not impose self-contact.

To enforce self-contact for all surfaces in the enclosing `CONTACT DEFINITION` block or for a subset of those surfaces specified in a `CONTACT SURFACES` line, the following command line must be present:

```
SELF CONTACT = ON
```

The default value for `SELF CONTACT` is `OFF`.

If general contact or self-contact is enabled, contact enforcement may be disabled for individual surfaces or sets of surfaces by using the `INTERACTION BEHAVIOR = NO_INTERACTION` line in specific `INTERACTION` command blocks.

8.6.1.2 Specific Surfaces Identification

```
SURFACES = <string list>surface_names
```

This command line identifies the contact surfaces to which the surface interaction values defined in the `INTERACTION DEFAULTS` command block will apply. The `surface_names` list can include any surface specified in a `CONTACT SURFACE` command line, a `CONTACT SURFACE` command block, or created automatically by a `SKIN ALL BLOCKS` command line in the enclosing `BEGIN CONTACT DEFINITION` scope.

In the absence of a `SURFACES` command line, the interaction defaults will apply to all contact surfaces defined in the enclosing `BEGIN CONTACT DEFINITION` scope.

8.6.1.3 Friction Model

```
FRICITION MODEL = <string>friction_model_name (FRICITIONLESS)
```

The `FRICITION MODEL` command line specifies how surfaces interact with each other. Several simple friction models can be used for `friction\_model\_name` including `FRICITIONLESS`, `TIED`, and `GLUED`. Alternatively `friction\_model\_name` can be the name of a more complex friction model defined elsewhere in the `CONTACT DEFINITION` command block. See [Section 8.5](#) for a listing of the available friction models.

The following set of commands will apply the Coulomb friction model `fric1` to all surface interactions.

```
BEGIN CONSTANT FRICITION MODEL fric1
  COULOMB COEFFICIENT = 0.2
END
```

```
BEGIN INTERACTION DEFAULTS
  SELF CONTACT = ON
  GENERAL CONTACT = ON
  FRICITION MODEL = fric1
END
```

If the `FRICITION MODEL` command is not present, the default interface model is frictionless contact.

8.6.1.4 Interaction Behavior

```
INTERACTION BEHAVIOR = <string>SLIDING|NO_INTERACTION (SLIDING)
```

The `INTERACTION BEHAVIOR` command line specifies whether contact interactions are active. By default, all the interactions specified in the `INTERACTION DEFAULTS` command block through the `GENERAL CONTACT`, `SELF CONTACT`, or `CONTACT SURFACES` commands are active, that is `SLIDING`. However, setting `INTERACTION BEHAVIOR` to `NO_INTERACTION` will deactivate contact for all these surfaces. In either case, the value for `INTERACTION BEHAVIOR` set in the `INTERACTION DEFAULTS` block can be overwritten with the use of the same command line in individual `INTERACTION` command blocks.

8.6.1.5 Constraint Formulation

```
CONSTRAINT FORMULATION = NODE_FACE|FACE_FACE
```

The `CONSTRAINT FORMULATION` command specifies either node/face or face/face enforcement for contact constraints. This command only applies to Dash contact because implicit kinematic contact and ACME only support node/face contact. With Dash contact, face/face contact is used by default except when one of the surfaces is explicitly defined by a node set. In that case, Dash

will automatically use node/face in which the “slave” surface will be the one defined by a set of nodes and the “master” surface will be the contact surface defined by faces. See Section 8.3 for defining contact surfaces by nodes or faces.

See Section 8.6.4 for more information on the difference between node/face and face/face constraints.



8.6.1.6 Augmented Lagrange Penalty

```
AL PENALTY = <real>normalPenalty(1.0) [<real>tangentialPenalty]
```

The AL PENALTY command is used to scale the penalty stiffness that is internally set based on the nodal stiffness of the parts in contact. Both normal and tangential penalty scale factors default to 1.0. If only on one value is specified, it will be used for both the normal and tangential penalty scale factors. When both normalPenalty and tangentialPenalty are entered, the normal penalty scale factor is set to normalPenalty, and the tangential penalty scale factor is set to tangentialPenalty.

8.6.2 Specific Interactions

```
BEGIN INTERACTION [<string>name]
  # surface identification
  SURFACES = <string list>surfaces [EXCLUDE <string list>surfaces]
  MASTER   = <string list>surfaces [EXCLUDE <string list>surfaces]
  SLAVE     = <string list>surfaces [EXCLUDE <string list>surfaces]
  #
  INTERACTION BEHAVIOR = <string>SLIDING|NO_INTERACTION(SLIDING)
  CONSTRAINT FORMULATION = <string>NODE_FACE|FACE_FACE
  FRICTION MODEL = <string>friction_model_name(FRICTIONLESS)
  INTERFACE MATERIAL = <string>int_matl_name MODEL = <string>int_model_name

  #
  # tolerances
  NORMAL TOLERANCE = <real>norm_tol
  TANGENTIAL TOLERANCE = <real>tang_tol
  OVERLAP NORMAL TOLERANCE = <real>over_norm_tol
  OVERLAP TANGENTIAL TOLERANCE = <real>over_tang_tol
  CAPTURE TOLERANCE = <real>cap_tol
  #
  # other implicit only commands
  AL PENALTY = <real>normalPenalty(1.0) [<real>tangentialPenalty]
  PUSHBACK FACTOR = <real>pushback_factor(1.0)
  TENSION RELEASE = <real>ten_release(0.0)
  TENSION RELEASE FUNCTION = <string>ten_release_func
  #
END [INTERACTION <string>name]
```

Values for specific contact interactions can be set using the INTERACTION command block. If an

INTERACTION DEFAULTS command block is present within a CONTACT DEFINITION command block, the values provided by an INTERACTION command block override the defined defaults. If an INTERACTION DEFAULTS command block is not present, only those interactions defined by INTERACTION command blocks are searched for contact, and values without system defaults must be specified.

The INTERACTION command block begins with:

```
BEGIN INTERACTION [<string>name]
```

and ends with:

```
END [INTERACTION <string>name]
```

where `name` is a name for the interaction. Note that this name is currently used only for informational output purposes and is not required.

The valid commands within an INTERACTION command block are described in Section [8.6.2.1](#) through Section [8.6.2.2](#). Section [8.14.3](#).

8.6.2.1 Surface Identification

```
# preferred for explicit and implicit face/face contact
SURFACES = <string list>surfaces [EXCLUDE <string list>surfaces]
#
# preferred for implicit node/face enforcement
MASTER   = <string list>surfaces [EXCLUDE <string list>surfaces]
SLAVE     = <string list>surfaces [EXCLUDE <string list>surfaces]
```

There are two methods to identify the surfaces described by a specific interaction. One method is to identify all surfaces in a single line with the SURFACES command line. The surface list must contain at least two surfaces. This command will define a symmetric two-way contact interaction between each surface on the command line and every other surface listed. To specify self contact, the surface must appear in the SURFACES command line twice.

The second method is to use the MASTER and SLAVE command lines. In Sierra/SM, contact surfaces must be identified using the MASTER and SLAVE command lines for kinematic enforcement. For augmented Lagrange enforcement, either method of identifying the surfaces may be used. The nodes of the slave surfaces are searched against the faces of the master surfaces. Each of these command lines takes as input a list of names of contact surfaces defined in the CONTACT DEFINITION block. A master slave interaction will be defined between each surface in the master list and each surface in the slave list and a surface may not be present in both lists.

The option `all_surfaces` is a special reserved word equivalent to directly specifying all contact surfaces known by the contact block in a list. Optionally, the EXCLUDE keyword can be used along with a list of surface names to exclude from the list.

If the SURFACES, MASTER, or SLAVE command lines appear multiple times within an INTERACTION block, their surface lists will be concatenated. The effect is equivalent to specifying all the surface names on a single line. The SURFACES command line cannot be used in the same INTERACTION block as MASTER and SLAVE commands.

The following examples demonstrate ways to identify contact surfaces involved in an interaction:

This command defines a symmetric interaction between *s1* and *s2*:

```
SURFACES = s1 s2
```

This defines a symmetric self contact interaction between *s1* and itself:

```
SURFACES = s1 s1
```

This defines a set of symmetric interactions between *s1* and *s2*, *s1* and *s3*, *s2* and *s3*:

```
SURFACES = s1 s2 s3
```

This defines a full set of symmetric interactions between *s1* and itself, *s2* and itself, and between *s1* and *s2*:

```
SURFACES = s1 s1 s2 s2
```

This command defines a set of interactions between all pairs of defined surfaces in the model, with the exception of surfaces *s7* and *s8*, which have no interactions:

```
SURFACES = all_surfaces exclude s7 s8
```

This command defines a set of interactions between all pairs of defined surfaces in the model, as well interactions between each surface and itself, with the exception of surfaces *s7* and *s8*, which have no interactions:

```
SURFACES = all_surfaces all_surfaces exclude s7 s8
```

These commands define a one-way interaction between the nodes of *s1* and the faces of *m1*:

```
MASTER = m1  
SLAVE = s1
```

These commands define a set of one-way interactions between the nodes of *s1* and the faces of both *m1* and *m2*, and between the nodes of *s2* and the faces of both *m1*, the nodes of *s2* and *m2*.

```
MASTER = m1 m2  
SLAVE = s1 s2
```

These commands define that the nodes of surface *s1* are slaved to all other contact faces in the contact definition block.

```
MASTER = all_surfaces exclude s1  
SLAVE = s1
```

8.6.2.2 Interaction Behavior

```
INTERACTION BEHAVIOR = <string>SLIDING|NO_INTERACTION(SLIDING)
```

This command sets the search behavior for a specific contact surface pair or for self-contact of a surface. See Section 8.6.1.4 for a discussion of this command line.

Since the default value is `SLIDING`, this command line is most often used to deactivate enforcement for the specific surfaces listed in this `INTERACTION` by setting the value to `NO_INTERACTION`.

8.6.2.3 Constraint Formulation

```
CONSTRAINT FORMULATION = NODE_FACE|FACE_FACE
```

The `CONSTRAINT FORMULATION` command specifies node/face or face/face enforcement for contact constraints for the specific surfaces listed in the `INTERACTION` block. See Section 8.6.1.5 for a discussion of this command line. See Section 8.6.4 for more information on the difference between node/face and face/face constraints.

8.6.2.4 Friction Model

```
FRICTION MODEL = <string>fric_model_name(FRICTIONLESS)
```

You can set the friction model for the interaction for a specific contact surface pair or for self-contact of a surface by using the above command line in an `INTERACTION` command block. See Section 8.6.1.3 for a discussion of this command line.

8.6.2.5 Interface Material

```
INTERFACE MATERIAL = <string>int_matl_name  
MODEL = <string>int_model_name
```

Any of the cohesive zone models available for use with interface elements (see Section 5.3) can be used as interface models within contact. Interactions between faces are created by a contact search performed in the initial configuration (similar to tied contact), so cohesive zones are only created between faces that are initially contacting. The relative motion of the faces on opposing sides of an interface is governed by the behavior of the cohesive zone model.

Note that these interface models can only be used with Dash contact. Additionally, due to the need to store state variables, these models can only be used on interfaces that use node-face contact interactions.

The stiffness of the compliant joint model changes over time based on the loading history of the joint. This stiffness is not explicitly taken into account by the critical time step estimate. Thus, if the critical time step of the contact joint is lower than the critical time step used by the analysis, the analysis may become unstable. If this occurs, the user will need to either manually decrease the integration time step used by the code or change parameters in this model to decrease the stiffness of the joint.

8.6.2.6 Tolerances

```
NORMAL TOLERANCE = <real>norm_tol
TANGENTIAL TOLERANCE = <real>tang_tol
OVERLAP NORMAL TOLERANCE = <real>over_norm_tol
OVERLAP TANGENTIAL TOLERANCE = <real>over_tang_tol
CAPTURE TOLERANCE = <real>cap_tol
```

 **Implicit**

Tolerances for the interaction of specific contact surfaces or for self-contact of surfaces are set with the above tolerance-related command lines in an `INTERACTION` command block. See Section 8.9.6.1 on search tolerances and Section 8.9.4 on overlap tolerances for a complete discussion of tolerances for contact.

The contact functionality in Sierra/SM uses a box defined around each face to locate nodes that may potentially contact the face. This box is defined by a tolerance normal to the face and another tolerance tangential to the face (see Figure 8.21). The code adds the maximum motion over a time step to these tolerances when identifying interactions. In the above command lines, the parameter `norm_tol` is the normal tolerance (defined on the `NORMAL TOLERANCE` command line) for the search box and the parameter `tang_tol` is the tangential tolerance (defined on the `TANGENTIAL TOLERANCE` command line) for the search box.

In most cases, the normal and tangential tolerances computed internally are optimal values, and manual setting of tolerances should be avoided. If specific normal and tangential tolerances are set, the tolerances used should not exceed roughly half of the minimum characteristic face size. Tolerances beyond that length may cause instability in the contact solution.

The `CAPTURE TOLERANCE`, which should be no larger than the `NORMAL TOLERANCE`, is used to determine which slave nodes near a master surface should be pulled to the master surface and considered for contact. (This applies to slave nodes which have not penetrated the master surface. Slave nodes that have penetrated the master surface will be pushed to the surface regardless of the `CAPTURE TOLERANCE`.) If later checks determine that a slave node that was pulled to the surface (because it was nearer the surface than the `CAPTURE TOLERANCE` value) is in fact in tension and should be released, that slave node will not be considered as a potential contact node again during the load step as long as the node remains within the `NORMAL TOLERANCE`.

 **Implicit Only**

8.6.2.7 Augmented Lagrange Penalty

```
AL PENALTY = <real>normalPenalty(1.0) [<real>tangentialPenalty]
```

See Section 8.6.1.5 for a discussion of this command line.

8.6.2.8 Pushback Factor

```
PUSHBACK FACTOR = <real>pushback_factor
```

Implicit contact feature.

The command line `PUSHBACK FACTOR` can be used to set the fraction of the gap to be removed in a contact model problem. The default value is 1.0 which removes the entire gap in one contact

 **Implicit Only**

model problem. Setting the pushback factor to 0.25 will result in the gap being removed in 4 contact model problems.



8.6.2.9 Tension Release

```
TENSION RELEASE = <real>ten_release(0.0)
```

The command line `TENSION RELEASE` can be used to set a traction threshold below which slave nodes that have come into contact with master faces will not be released. When this value is set and a slave node is in tension, it will only be allowed to pull away from the master surface if the slave node's traction is greater than the `TENSION RELEASE` tolerance. The default value for `TENSION RELEASE` is 0.0.



8.6.2.10 Tension Release Function

```
TENSION RELEASE FUNCTION = <real>ten_release_func
```

The command line `TENSION RELEASE FUNCTION` provides a way for the tension release threshold to be set using a function from the input file. If the `TENSION RELEASE` line command is also present, the threshold used by the code will be the `TENSION RELEASE` value multiplied by the value obtained from the function.

8.6.3 Interaction Behavior for Particle Element Blocks

Element blocks containing particle elements (SPH, Mass Particle, peridynamics, or point masses) undergo special treatment, as described below.

Because the SPH and peridynamics element algorithms already define contact like interactions elements, it is typically not desirable to enforce self-contact within those element blocks, so self-contact is disabled by default for these blocks. For the same reason, contact between blocks of SPH elements is disabled by default.

Mass particles define both self-contact and inter-block interactions by default. Point masses behave similar to mass particles, but the 'skin' of a point mass block is just the nodes of the block. The 'skin' of a mass particle block is lofted spheres. Node/face contact for other particle types can be used by defining contact on the particle node sets, using the 'contact nodeset' command, or specifying the constraint formulation to `NODE_FACE`.

Elements converted to particles are placed in a new element block. The name of the new element block is the name of the parent block with `_particles` appended. For example, particles created from `block_1` are placed in a new element block named `block_1_particles`. The `SKIN ALL BLOCKS` command creates a new contact surface containing the particles with the same name as the particle block (`block_1_particles` in this example). By default, the contact surface for the particles has the same interactions as does the parent block, subject to the exceptions listed above for interactions between particle blocks.

Any of the above defaults can be overridden by defining individual interactions. For example, if `block_1` and `block_2` are SPH particle blocks, they will not interact though contact by default.

However, the following command block can be included to cause contact to be enforced between them.

```
BEGIN INTERACTION
  SURFACES = block_1 block_2
END
```

If a single contact surface is defined by combining multiple types of objects, the special particle-specific default behavior described in this section is disabled. For example, the following command block will turn on self-contact for all entities within the surface `s1`, including between the particles in `block_2_particles`.

```
BEGIN INTERACTION DEFAULTS
  SELF CONTACT = ON
END
CONTACT SURFACE s1 CONTAINS block_1 block_2_particles surface_3
```

8.6.4 Face/Face vs. Node/Face constraints

Interactions can be formulated as either face/face constraints or node/face constraints. Face/face is the default for all surface on surface interactions. Node/face is the default of node set on surface interactions or certain types of particle on surface interactions. The constraint formulation type can be set in the `INTERACTION` or `INTERACTION DEFAULTS` command blocks. The two types of constraints have trade-offs in terms of robustness and accuracy discussed in this section.

A node/face constraint is a constraint between a single node and a single face. In a node/face constraint a node is not allowed move through a face. The node is tied to a specific spot on the face, allowed to slide along the face, or allowed to move away from the face depending on the friction model used. An example of node for surface nodes (red) sliding off of a block (blue) is shown in figure 8.14. In the figure the constraints themselves are indicated with green arrows.

Node/face constraints have some limitations. In order to correctly resolve nodes sliding around corners of curved surfaces node/face constraints generally require use of some tangential tolerance (shown in dashed line). Note that with node face/face constraints the red nodes of the constraints may not penetrate the blue faces. The blue nodes however, may penetrate the red faces. Node/face constraints can potentially be setup symmetrically so that both sides of the interactions contain both the nodes and the faces of constraints. However, this symmetric setup leads to chaining of constraints that may be difficult to resolve particularly in implicit analyses.

A face/face constraint is a constraint between two faces. The face/face constraint constrains the overlapping area of the two faces. In a face/face constraint no part of either face is allowed to move through any part of the other face. Faces remain tied together, can slide against one another, or move apart depending on the friction model used. An example of a surface (red) sliding off of a block (blue) with face/face constraints is shown in Figure 8.14. In the figure the constraints are represented with green regions of overlap between faces.

Face/face constraints do not require any tangential tolerances, though do have some limitations. Face/face constraints are most accurate when the two constrained constrained faces are parallel.

Face/face constraints can unaligned faces, though sometimes with accuracy limitations particularly for frictional quantities. Face/face constraints are generally symmetric by nature with no distinct master or slave sides of the interaction.

When tying together meshes with node/face constraints the forces are applied to discrete points potentially resulting in artificial stress concentrations. When tying with face/face constraints the contact forces are instead applied via tractions on the constrained faces. Thus face/face constraints generally results in a smoother tied solution with less severe artificial stress concentrations.

For node/face constraints the friction law is evaluated and the forces applied directly at the slave nodes of the constraints. This has the advantage that frictional output quantities can be accumulated easily over time at the nodal points. For face/face constraints the exact location of the constraint moves in time. This can smear the history of frictional quantities making them less distinct and less accurate.

In some cases, face/face constraints are more robust than node-/ace constraints and can more accurately solve certain classes of problems. Face/face constraints often work better than node/face for problems with high loading rates and stiffening materials. Node-/face contact enforcement tends to aggravate hourglass modes when used on these types of problems. Additionally, problems with many blocks meeting at corners may often run more robustly with face/face constraints. In node/face constraints a node will only interact with one other face at a time making corner contacts inherently ambiguous. For face/face constraints the ambiguity is removed by moving the constraints to faces where the interaction is unambiguous even for very complex geometry.

Node/Face constraints are generally a bit faster to evaluate than face/face constraints, though the performance difference is usually in most production sized analyses.

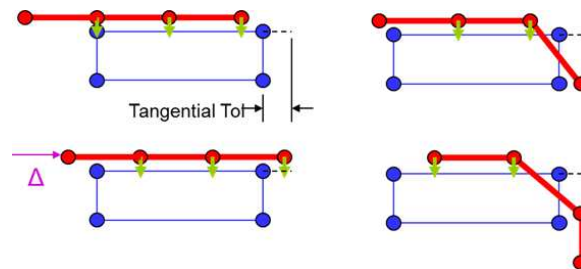


Figure 8.13: Node/Face Constraint example

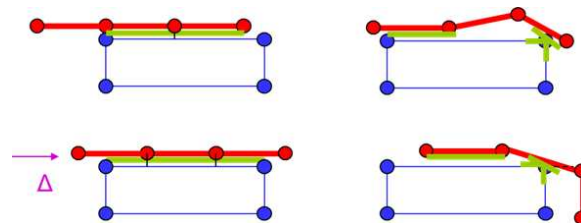


Figure 8.14: Face/Face Constraint example

8.7 Active Periods

```
ACTIVE PERIODS = <string list>period_names
```

A `CONTACT DEFINITION` is active for all periods by default. The contact definition can be activated for specific time periods with the `ACTIVE PERIODS` command line (See Section [2.6](#)).

8.8 Contact with Analytic Objects

Contact with analytic objects is desired if a user would like to enforce contact on their meshed body but not modify the mesh file to create the object to come into contact with. This object can be created with the below syntax:

```
BEGIN ANALYTIC OBJECT <string>analytic_object_name
  TYPE = <string> PLANE
  NORMAL DIRECTION = <real>x_dir <real>y_dir <real>z_dir
  ORIGIN = <real>x <real>y <real>z
  SCALE FACTOR = <real> scale_factor
END [ANALYTIC OBJECT]
```

This syntax should be placed in the `REGION` command scope.

When using an analytic object, it is required to give your analytic object a name. This name is then used when specifying contact surfaces. For example, if you were to name your analytic object "analytic_obj1", you would specify the contact surface:

```
CONTACT SURFACE c1 CONTAINS analytic_obj1
```

The rest of the contact definition is defined using the usual commands. See section Section 8.3 for details.

The input command `TYPE` specifies the type of analytic object to create. Currently the only option is to create an analytic plane.

The input command `NORMAL DIRECTION` specifies the normal direction of the plane to be created.

The input command `ORIGIN` specifies the origin at which the plane is created.

By default, the plane that is created is 3X the size of the maximum length of the meshed body. If it is desired to make the plane a different size, the input command `SCALE FACTOR` can be used to scale the size of the plane.

8.8.1 Kinematic BC's with Analytic Objects

A kinematic boundary condition (such as fixed displacement, prescribed displacement, etc.) can be applied to the analytic object that is created. Within the boundary condition definition, the block name or surface name should be the name given to the analytic object.

If no kinematic BC is defined on the analytic object, a fixed displacement BC will be applied by default.

Other boundary conditions such as pressure, gravity, or force will not function on analytic planes.

8.9 Contact Algorithm Options

8.9.1 Contact Library

```
SEARCH = ACME|DASH(DASH)
```

Dash and ACME are two separate search and enforcement algorithms for defining explicit dynamics or implicit contact. Generally any existing ACME or Dash contact command block may be converted to a Dash or ACME command block simply by using this one command. However, Dash will ignore some ACME options and ACME will ignore some Dash options.

8.9.1.1 How Dash and ACME differ

Dash offers a facet-based enforcement algorithm. By default, Dash enforces face-face constraints. Both ACME and Dash offer a node-face based enforcement algorithm.

Dash is designed to work correctly for a wide variety of problems with minimal user intervention. It is generally recommended that Dash be used with a minimal set of inputs, for example `skin all blocks`, `self contact = on`, and/or `general contact = on`, and definition of the needed interaction friction models. Dash ignores tangential tolerances for face/face constraints. For more details (see discussion in Section 8.9.1.2).

For explicit contact, Dash uses an iterative augmented Lagrange stiffness enforcement scheme. Dash enforcement explicitly balances momentum by always applying equal and opposite forces. In contrast, ACME uses an iterative kinematic enforcement scheme. The kinematic enforcement scheme can more exactly remove gaps, but may not exactly satisfy force balance at a contact interface. With sufficient iterations, both the Dash and ACME enforcement schemes will converge to the same results.

For implicit contact, the Dash contact search can be used with either kinematic or augmented Lagrange enforcement. When Dash is used with the implicit kinematic enforcement algorithm, only node-face interactions are enforced, and the default behavior is changed from face-face to node-face.

Dash represents lofted structural elements differently than ACME. Dash can only represent shells via lofted geometry and has no option to treat shells as if they have no thickness for contact. Dash adds the edge faces to lofted shells, allowing edge and edge to face contact to function correctly on shell elements. Dash also handles beam on beam contact via lofted geometry. Beam objects are turned into representative hexagonal prisms that have roughly the same cross sectional area that the beam elements have. Beam contact is functional for edge-on or end-on cases.

These differences in the way structural elements are treated for contact make Dash significantly more capable than ACME in this area. ACME cannot treat any contact between shell edges and other entities, and cannot handle any contact with beam edges.

8.9.1.2 Current Dash Usage Guidelines

Use Dash for remeshing problems. ACME does not perform well on changing topologies. Dash is more robust on problems with element death. ACME often must artificially kill a large number

of additional elements other than those which have reached a user defined death criteria. ACME requires these additional element removals to work around certain surface topology restrictions. Dash has no topology restrictions so does not need to kill any extra elements to maintain a valid contact topology.

When performing contact involving shells, beams, and particles, Dash turns the elements into lofted volumes. ACME is able to loft shells and particles, but cannot loft beams.

Both ACME and Dash support basic friction models (tied, Coulomb friction, frictionless, etc.) However, some other friction models (spring weld, surface weld, area weld, adhesion, junction, threaded, pv_dependent, and user subroutine) are not yet available for use with Dash.

Prefer using block skinning with Dash, and avoid defining contact surfaces based on side sets. Dash uses ray tracing algorithms to determine if certain contact points are inside or outside of bodies. If the surfaces used for contact with Dash do not define fully enclosed bodies, it is difficult for the algorithm to determine whether a point is inside or outside a body.

Dash accepts the same surface definitions as does ACME. Dash also uses identical syntax for interaction definition.

8.9.2 Dash Specific Options

```
BEGIN DASH OPTIONS
  INTERACTION DEFINITION SCHEME = EXPLICIT|AUTOMATIC (AUTOMATIC)
  SEARCH LENGTH SCALING = <real>scale(0.15)
  TIED SEARCH LENGTH SCALING = <real>scale(0.25)
  ACCURACY LEVEL = <real>accuracy(1.0)
  SUBDIVISION LEVEL = <int>sublevel(0)
  QUAD PROJECTION TYPE =
    TRIANGULAR_FACETS|QUAD (TRIANGULAR_FACETS)
  HIDDEN SELF CONTACT POSSIBLE = FALSE|TRUE (TRUE)
  MAX CONTACT SUB STEPS = <int>maxSteps(100)
  NODE NORMAL SMOOTHING = AREA_WEIGHTED|ANGLE_WEIGHTED (ANGLE_WEIGHTED)
END
```

The `DASH OPTIONS` command block contains a few options unique to Dash. Generally Dash is designed to run accurately with as few user specified parameters as possible, however, some extra options are available. Note, the options in this block should generally be used only by experienced analysts for specific purposes.

The `INTERACTION DEFINITION SCHEME` command controls how contact interactions are defined. By default most Dash interactions are one sided. As the full volume of overlap is removed there is no concept of master face or slave face. By default Dash automatically picks an optimal enforcement order for each interaction pair based on face size. However, the ability exists to explicitly use the interactions defined in the input deck by supplying the `EXPLICIT` option to this command. This may allow better control of interaction setup in some circumstances or can be used to force Dash to use a symmetric interaction when it would by default use a one sided interaction.

The `SEARCH LENGTH SCALING` how Dash computes automatic contact tolerances. The automated search length used by contact at each face is the characteristic size of the face times this

value. Note that the default value of this command should be used in almost all circumstances. One use of this command is to zero the automatic tolerances to force the contact algorithm to use the user provided search tolerances (normally contact uses the larger of the user provided or the default search tolerance). In some cases such as high velocity contact where objects move further than the default search length over a single step use of a larger factor can be helpful. Note though that use of a scaling factor larger than 0.50 can cause instability in the contact solution and should be avoided.

Tied interactions used the a length scaling given by `TIED SEARCH LENGTH SCALING`, by default tied interaction use a somewhat larger search length. If a tied interaction is initially further than the characteristic face size times the search length scaling tied contact will not be found. A larger or smaller value for this scaling number can be provided. However, values larger than 0.50 should be avoided as they can result in instability.

The `ACCURACY LEVEL` command provides a single dial to change various numerical parameters in the code related to the trade off between algorithmic computation time and algorithm accuracy. The default value is one, large values will tend to increase accuracy while increasing computation time and lower values will tend to decrease accuracy while decreasing computation time. A typical relevant range of this parameter is from 0.1 to 100.0.

At higher levels of accuracy the contact algorithm will allow less material interpenetration to occur, provide more accurate iterations of frictional forces, and be less sensitive to numerical noise and error. Setting the accuracy level parameter will also affect the `MOMENTUM BALANCE ITERATIONS` and `NUM GEOMETRY UPDATE ITERATIONS` contact enforcement parameters.

The `SUBDIVISION LEVEL` command provides a means to force a triangular face to be subdivided. Each quadrilateral face is originally split into four triangular faces. A subdivision level splits each tri-face edge into `sublevel + 1` segments. An increased subdivision level give a more accurate solution because the overlap volume is more accurately calculated. This increased accuracy, however, comes with a significantly increased computational cost.

The `QUAD PROJECTION TYPE` command provides a means to select the projection scheme of contact interactions to potentially warped quad surfaces. The default `TRIANGULAR_FACETS` option computes the projection point by dividing each quad into four flat triangles. The `QUAD` option computes the true position of the projection point on the warped quad. The `TRIANGULAR_FACETS` is faster to compute and is more robust in loading where deformations and potential warping is large. The `QUAD` option is somewhat more expensive to compute but can give a more consistent numerical result in an analysis where deformation is small.

The `HIDDEN SELF CONTACT POSSIBLE` command is used to control how Dash automatically sets up interactions. Self contact interactions use special more expensive contact logic. This more complex logic is needed to resolve constraint chaining issues that can happen in self contact. If two surfaces share (say contiguous blocks) and those two surfaces contact then by default the contact logic is applied as well as the same constraint chaining issues arise. The `HIDDEN SELF CONTACT POSSIBLE` command can be used to turn off this more expensive self contact logic. This can sometimes improve performance but may introduce instabilities in the contact solution.

In high velocity contact situations explicit contact enforcement may subdivide the time step to help accurately capture interactions. The `MAX CONTACT SUB STEPS` can be used to set the maximum

number of sub steps taken over a step. In practice the only use of this command would be to drop number of sub steps taken to one in a problem that is encountering some sort of issue with the sub stepping computations.

8.9.2.1 Node Normal Smoothing

```
BEGIN DASH OPTIONS
  NODE NORMAL SMOOTHING = AREA_WEIGHTED | ANGLE_WEIGHTED (ANGLE_WEIGHTED)
END
```

The `NODE NORMAL SMOOTHING` affects how the nodal normals are computed for Dash node/face contact. The node normal is used to determine which face a given node will contact. In cases where multiple faces are potential contact candidates the node will contact the face with the face normal that most opposed the node normal.

There are two options for normal smoothing. The default `ANGLE_WEIGHTED` option computes the nodal normal as the sector angle weighted normals of attached faces. Consider the example shown in Figure 8.15 two faces come together at right angles. The sector angle (the angle at the face corner that attaches to the node) of both faces is 90 degrees. Thus the two faces contribute equally to the node normal which is at 45 degrees. The `AREA_WEIGHTED` option computes the nodal normal as the face angle weighted normals of attached faces. In Figure 8.15 the top face has more area than the side face. This pulls the node normal more in the direction of the top face.

The `ANGLE_WEIGHTED` is generally more accurate and has several beneficial property. With the angle weighted normal the node normal at edges and corners is not dependent on the specific mesh or element shapes. This is a very useful property for convergence studies. Additionally for any convex geometry the angle weighted normal is guaranteed to be at least partially aligned with all attached face normals. This can make finding corner node/face interactions more robust. The primary advantage of the `AREA_WEIGHTED` option is speed. Use of the `AREA_WEIGHTED` option can improve runtime by up to 20% in some cases. Additionally the `AREA_WEIGHTED` option was the Sierra/SM default behavior prior to version 4.46 and is maintained for backward compatibility.

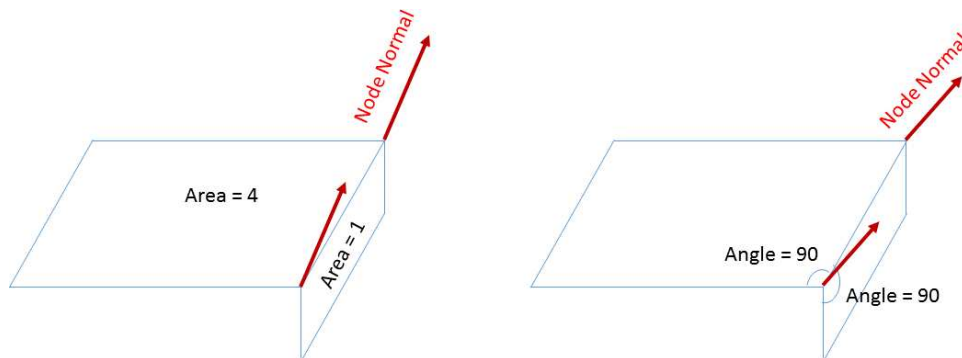


Figure 8.15: Node Normal Computation Options



8.9.3 Enforcement

```
ENFORCEMENT = <string>AL | KINEMATIC (AL)
```

```
CONTACT RESIDUAL NORM SCALE FACTOR = <real>scale
```

The `ENFORCEMENT` command line controls whether kinematic or Augmented Lagrange (AL) enforcement will be used in the current `CONTACT DEFINITION` block.

The `AL` option enforces contact using an Augmented Lagrange algorithm. This option can be used only in conjunction with the Dash search, which is enabled using the `SEARCH = DASH` command (See Section 8.9.1). and is the default option when using the Dash search. The `FRICTION MODEL` commands are necessary with this command and varying contact models can be enforced within the same `CONTACT DEFINITION` block.

If kinematic enforcement is enabled (by using the `KINEMATIC` option), the enforcement algorithm and the friction model are linked, and only one friction model can be used per `CONTACT DEFINITION` block. If a combination of tied, frictionless, and frictional contact is to be enforced, multiple `CONTACT DEFINITION` blocks must be included in the input file.

The `CONTACT RESIDUAL NORM SCALE FACTOR` Applies scale factor to contact residual norm of the contact definition in which this command is embedded. This allows a specific contact definition to contribute with different weight factor to the total contact residual norm. For example, one may want the contact residual from `ARS` to be the only contribution to the contact residual norm. This can be achieved by suppressing all other contributions using a scale factor of 0. The default value is unity.

8.9.4 Remove Initial Overlap

```
INITIAL OVERLAP REMOVAL = <string>ON|OFF(OFF)
```

```
BEGIN REMOVE INITIAL OVERLAP
```

```
    OVERLAP NORMAL TOLERANCE = <real>over_norm_tol
```

```
    OVERLAP TANGENTIAL TOLERANCE = <real>over_tang_tol
```

```
    OVERLAP ITERATIONS = <integer>iterations(100)
```

```
    DEBUG ITERATION PLOT = <string>ON|OFF(OFF)
```

```
    REMOVE OVERLAP AT START OF TIME PERIODS = <string list>periods
```

```
END REMOVE INITIAL OVERLAP
```

```
OVERLAP REMOVAL ONLY = OFF|ON(OFF)
```

Meshes supplied for finite element analyses often have some level of initial mesh overlap, where finite element nodes reside inside the volume of nearby elements. In particular, this will happen when meshes meet at non-contiguous curved boundaries.

In explicit analyses the contact algorithm will attempt to push the overlapped constraints to zero gap. If a mesh has initial overlap contact will attempt to push out that overlap over the first explicit time step. As the explicit time step is often small, this may result in an extremely large force, velocity, and energy input into the initial mesh. Additionally, these initial forces are unphysical and may produce erroneous stress waves. Initial overlap removal is a mechanism for modifying the initial mesh to eliminate any initial overlap of contact surfaces. This is done before taking the first

explicit dynamics time step to avoid the aforementioned issues. The overlap removal algorithm is by default turned off, but can be turned on by including the `INITIAL OVERLAP REMOVAL = ON` command line or use of the `BEGIN REMOVE INITIAL OVERLAP` block. The `REMOVE INITIAL OVERLAP` command block can also be used to modify the default overlap removal settings described later in this section.

For an example of overlap removal, see Figure 8.16.

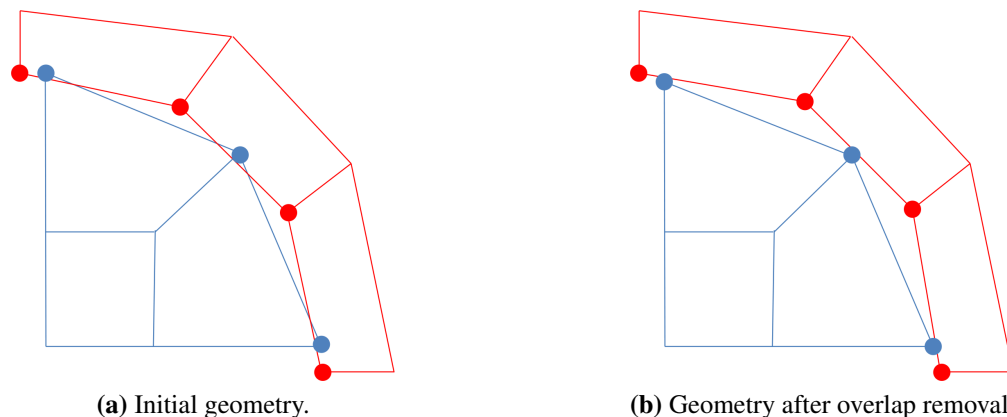


Figure 8.16: Effect of overlap removal on model geometry.

The process used to remove the initial overlap for three-dimensional solid elements involves changing the original coordinates of nodes on contact surfaces. Moving these coordinates yields a new mesh with the overlap removed; the overlap removal adds no initial stresses. Normal and tangential tolerances specific to overlap removal may be specified by the user for all contact surfaces in the `REMOVE INITIAL OVERLAP` command block. It is also possible to specify overlap normal and tangential tolerances on each surface pairing separately by using the `INTERACTION` command block. Overlap removal tolerances specified in `INTERACTION` command blocks will overwrite the tolerances specified in the `REMOVE INITIAL OVERLAP` command block. See Section 8.6.2 for details. The `REMOVE INITIAL OVERLAP` command block only removes overlaps that are detected among the surfaces and elements defined for contact, not all surfaces in the mesh. Overlap removal tolerances will be set automatically if no specific tolerances are given.

Overlap tolerances are used to designate a maximum amount of overlap to search for and remove. If the amount of overlap in the mesh is larger than those tolerances, then the overlap will not be found and removed.

The mesh modification done by the overlap removal feature changes the meshed geometry; therefore, also changing the mass and time step of affected elements. The modified mesh is returned in the results file and should be checked to ensure that the modifications are acceptable.

8.9.4.1 Overlap Removal Diagnostics

The log file will contain a listing for maximum remaining overlap detected in the model after each overlap iteration. This maximum remaining overlap number should drive to zero over the overlap removal iterations. The code will output a warning if any initial overlap still exists after the overlap removal algorithms complete all iterations. Unremoved initial overlap may still exist if there was

excessive overlap in the initial configuration that could not be removed without inverting elements, or if there exists some pathology in the model geometry that the overlap removal algorithm cannot repair.

The log file contains two tables that relate to overlap removal. The first table lists the maximum amount any node of each block has been moved as a result of overlap removal. This quantity can be viewed on the mesh on a node by node basis by plotting the `REMOVED_OVERLAP` nodal variable on the results file. The second table lists the approximate maximum amount of overlap that exists in any element at the end of the overlap removal process. This variable can be viewed on the mesh on an element basis by plotting the `OVERLAP_VOLUME_RATIO` element variable on the results file.

Note the remaining overlap value reported in the overlap removal iterations is computed in a slightly different way than the final overlap reported by the overlap volume ratio table. The overlap iteration number is based on surface intersections and nodal coordinates. The overlap volume ratio number is based on block volumes. The overlap volume ratio will detect overlaps such as whole volumes covering the same space that are not detected by the surface based algorithm. However, the overlap volume ratio also treats edge on edge overlaps and warped faces slightly differently than the surface based algorithm. As a result, small overlap volume ratios may be reported in a mesh even after the overlap removal algorithm completes successfully. If the overlap iterations drive the reported overlap to zero and the overlap volume ratio is some small number (generally one percent or lower) then the overlap removal algorithm was successful and all overlap was removed. Also note that the overlap removal algorithm will not remove overlap between bodies not in contact. The overlap volume ratio report will however still report overlaps for such bodies. Overlap of non-contacting bodies usually will not cause issues for models but is still reported to the log file for completeness.

8.9.4.2 Overlap Removal Commands

The `OVERLAP ITERATIONS` sets the number of overlap iterations that will be performed. The default value for `OVERLAP ITERATIONS` is 100 and is usually sufficient to remove overlap. More iterations may be able to remove more overlap. However, if the overlap cannot be completely removed in the default 100 iterations, it is likely that there is some model geometric pathology that cannot be resolved and additional iterations are unlikely to help. Overlap removal iterations may also be used after particle insertion, remeshing, geometric disconnection, and other operations that modify the mesh topology. A high allowed iteration count can make these operations more expensive. An alternative use of the `OVERLAP ITERATIONS` command is to visualize initial input mesh overlap to help detect potential meshing or merging errors. If `OVERLAP ITERATIONS` is set to 0 overlap diagnostics will be computed but no overlap will be removed. Thus with zero overlap iterations output variables such as element `overlap_volume_ratio` will show the overlap metrics found in the initial unmodified mesh.

The `DEBUG ITERATION PLOT` command line allows visualization of the overlap removal iterations. Turning the command line on will print each overlap removal iteration to the results output file. The nodal variable "removed_overlap" can be used to show the progress of each overlap removal iteration. If the `DEBUG ITERATION PLOT` command is used after overlap removal, the analysis will terminate. Consequently, this command should only be used for debugging overlap

removal.

The `REMOVE OVERLAP AT START OF TIME PERIODS` command causes the overlap removal to be done at the start of each of the listed time periods. By default, overlap removal is done only at the start of the first time period. This command can be used to remove small overlaps between the pieces which are in contact before starting a new analysis time period.



Warning: Do not use `REMOVE OVERLAP AT START OF TIME PERIODS`, when having only one time period in your input file. In this case, this command is not needed and might cause unexpected behavior due to additional unnecessary overlap iterations.

If the `OVERLAP REMOVAL ONLY = ON` is used, the contact definition will only be used to remove initial overlap from the mesh. No contact enforcement will take place after overlap removal. This option can be used if it is desired for the overlap removal operations to use a different contact setup from the friction contact enforcement which is defined in a separate `BEGIN CONTACT DEFINITION` block.

8.9.4.3 Remove Initial Overlap Usage Guidelines

The following is a set of guidelines and restrictions which apply to overlap removal:

- Overlap removal works only on the nodes which are part of the contact surfaces. This means overlap removal generally cannot successfully remove large amounts of overlap. Attempts to remove overlap of greater than half an element length may lead to inverted or poorly shaped elements, making further analysis impossible. In such cases, the large overlap needs to be removed manually by remeshing.
- It is recommended to use the default normal and tangential tolerances for overlap removal whenever possible.
- When working with lofted elements (shells, particles and beams) overlap removal may move the nodes of these elements, but will not change the lofted thickness of the elements.
- Overlap removal only removes overlaps, it does not close initial gaps. Overlap removal attempts to bring overlapping surfaces exactly into contact but in some cases may result in small initial gaps between parts.
- Overlap removal will not move nodes that are kinematically constrained (such as nodes with fixed displacement boundary conditions).
- When removing a gap at an interface, overlap removal will generally move both sides of that interface by equal amounts.
- A tied contact friction model will work without overlap removal. However, it is still preferable to use overlap removal with tied contact as exactly matching interfaces lead to more accurate constraints.
- When restarting an analysis, overlap removal is not applied to the restarted mesh. It is assumed that the restarted mesh is already in the proper configuration for contact.

8.9.5 Lofted Surface Options

Elements such as shells, beams, and particles are represented for contact with lofted geometry, which defines a geometric volume around these lower dimensional elements. This volume allows for contact to be unambiguously evaluated, so that situations such as stacked shells or edge on edge contact of shells can be handled naturally. Block skinning creates lofted geometry by default out of all shell, beam, and particle element blocks. Certain features of the lofted geometry may be controlled with the `SURFACE OPTIONS` command block.

```
BEGIN SURFACE OPTIONS
  SURFACE = <string list>surface_names
  REMOVE SURFACE = <string list>removed_surface_names
  BEAM RADIUS = <real>beam_radius
  LOFTED BEAM SHAPE = HEXAGON|OCTAGON|SQUARE (HEXAGON)
  PARTICLE RADIUS = <real>part_radius
  LOFTED PARTICLE SHAPE = CUBE|ICOSAHEDRON|OCTAHEDRON|TETRAHEDRON|
    PARENT_ELEMENT (ICOSAHEDRON)
  CONTACT SHELL THICKNESS =
    ACTUAL_THICKNESS|LET_CONTACT_CHOOSE (ACTUAL_THICKNESS)
  ALLOWABLE SHELL THICKNESS TO ELEMENT SIZE RATIOS =
    <real>lower_bound(0.1) TO <real>upper_bound(1.0)
  SHELL CONTACT LOFTING FACTOR = <real>loftFactor
  SHELL CONTACT THICKNESS = <real>thickness

  COINCIDENT SHELL TREATMENT = <string>
    DISALLOW|IGNORE|BEST (BEST)
  COINCIDENT SHELL HEX TREATMENT = <string>
    DISALLOW|IGNORE|TAPERED (TAPERED)
END [SURFACE OPTIONS]
```

The `SURFACE` and `REMOVE SURFACE` command lines specify which contact surfaces this command block refers to. These command lines are optional; by default the commands in the surface options block apply to all contact surfaces. If the `SURFACE` command line is used by itself, only those surfaces listed will be affected. If the `REMOVE SURFACE` command line is used by itself, all contact surfaces except those listed will be affected. Note, the surface names used in this command block are the names of the contact surfaces, not the names of the sidesets or blocks used to form those contact surfaces.

8.9.5.1 Lofted Beams

By default, contact represents the cross section of lofted beam elements by a six sided prism. The radius of the vertexes of the hexagons is the square root of the beam cross sectional area divided by pi. That is, the beam cross sectional area is converted into a circle, and then the hexagon end cap of the beam roughly represents this circle. Note that for a circular beam, the contact representation of that beam will somewhat match the actual beam geometry. However, for an I-Beam cross section, the shape of the contact representation may be significantly smaller than the actual cross section length scales. The representation of various beams in contact is shown in Figure 8.17.

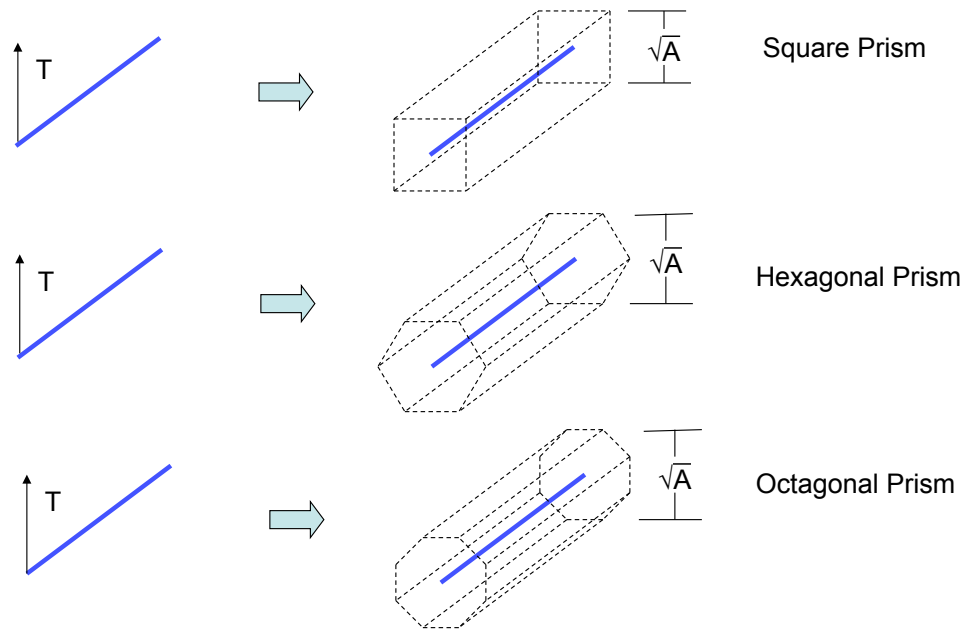


Figure 8.17: Lofted beam geometry

If a `T AXIS` command is given in a beam section the orientation of the lofted contact beam prism will be informed by that `t` axis. Specifically one of the flat sides of the prism will be placed perpendicular to the beam `t` axis. If no `taxis` is given in the beam section the orientation of the beam prism will be somewhat arbitrary.

Lofted beam elements are used to represent beams, springs, dampers, and other one dimensional elements.

The `BEAM RADIUS` command can be used to explicitly set the radius of the lofted beam elements, overriding the default use of the beam's cross sectional area.

The `LOFTED BEAM SHAPE` command changes how the beam is represented. Objects with fewer facets tend to be less accurate, but may run faster.

8.9.5.2 Lofted Particles

The representation of various particles in contact are shown in Figure 8.18. Lofted particle elements are used to represent mass particles, SPH particles, Peridynamics particles, and other zero dimensional point elements.

The `PARTICLE RADIUS` command can be used to explicitly set the radius of the lofted particle elements. By default, lofted particles are represented as twenty-sided icosahedra having radii equal to half of the value of the element `sph_radius` field.

The `LOFTED PARTICLE SHAPE` can change how finely the particle sphere is represented. Objects with fewer facets tend to be less accurate, but may result in improved run times. Note, the `PARENT\_ELEMENT` option is experimental and only available for hexahedral elements converted to particles at initialization.

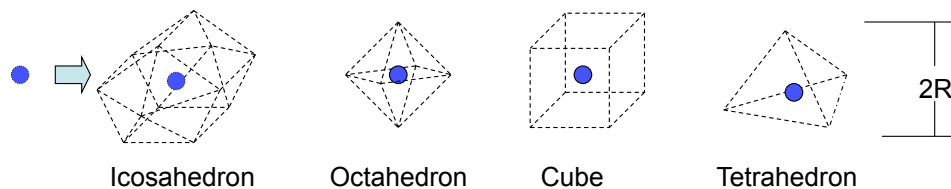


Figure 8.18: Lofted particle geometry

8.9.5.3 Lofted Shells

Lofted shells are represented by prisms around each shell element. An example representation of shells in contact is shown in Figure 8.19. Lofted shells are used to represent shells, membranes, and other two dimensional elements. The lofted shell geometry is also based on the element section `LOFTING FACTOR` as shown in Figure 6.3.

The `CONTACT SHELL THICKNESS` command line controls whether lofted shell contact uses the actual thickness of shell elements or a pseudo thickness that is computed by the contact algorithm. The default option to this command, `ACTUAL_THICKNESS`, causes the actual element thickness to be used. The `LET_CONTACT_CHOOSE` option to the `CONTACT SHELL THICKNESS` command tells the contact algorithm to create lofted geometries of the shells that are more appropriate for contact.

Shell elements that are very thick or very thin in relation to their in-plane dimension can be problematic in contact. A shell is considered very thin when the shell thickness is less than ten percent of the shell element's in plane size. Thin shells may require a larger number of sub-steps in the contact algorithm to ensure that objects do not pass completely through the shell in a single step, causing contact to be missed. For thick shells, particularly near corners, the exterior geometry of these prisms may become jagged and not representative of the actual shell geometry. A shell may be considered very thick when the shell thickness is greater than the shell element's in plane size.

Shell elements that are thick in relation to their in-plane dimension can produce ill-defined geometries, as seen in Figure 8.20. The `LET_CONTACT_CHOOSE` option can help alleviate issues related to thick or thin shells. By default, when the `LET_CONTACT_CHOOSE` option is used, lofted geometry thicknesses are ensured to be within 0.1–1.0 times the in-plane dimension of the shell element. If the contact-appropriate shell thickness is computed to be outside of these bounds, the lofted shell thickness will be set to the upper or lower bound as appropriate.

The permissible bounds for the ratios of contact-appropriate thickness to actual thickness can be set using the `lower_bound` and `upper_bound` parameters in the `ALLOWABLE SHELL THICKNESS TO ELEMENT SIZE RATIOS` command. The `ALLOWABLE SHELL THICKNESS` command is functional only for Dash contact.

A contact specific lofting factor for shells in can be set with the `SHELL CONTACT LOFTING FACTOR` command. For a description of shell lofting conventions see Section 6.2.4. By default contact uses the same lofting factor as does the shell element mechanics to loft the volume of the shell away from the shell element. The `SHELL CONTACT LOFTING FACTOR` will change the lofting factor for the shell in the contact algorithm, but not affect the lofting factor used for the element

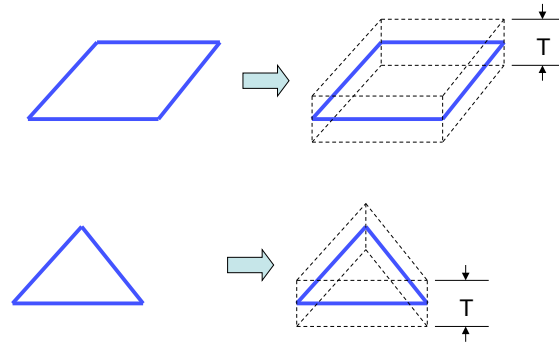


Figure 8.19: Lofted shell geometry

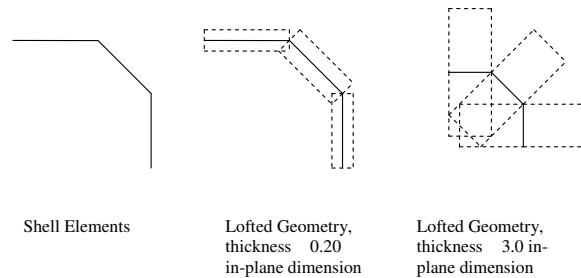


Figure 8.20: Example lofted geometries produced by shell lofting

mechanics. This contact shell lofting factor can be used to properly fit together shell meshes without the algorithmic time step and stability limitations that can come from the lofting of the element mechanics.

A contact specific thickness for shells can be set with the `SHELL CONTACT THICKNESS` command. By default contact uses the same thickness as does the shell element mechanics. Note, this command overrides the `ALLOWABLE SHELL THICKNESS TO ELEMENT SIZE RATIO` option and explicitly sets an absolute contact thickness.

8.9.5.4 Coincident Face Options

The `COINCIDENT SHELL TREATMENT` and `COINCIDENT HEX SHELL TREATMENT` command line identifies how elements that share the same nodes should be treated. This includes stacked shells and shells cladded onto the faces of the hexahedral element. Note as Dash has more robust shell contact logic by default Dash contact allows coincident shells while acme disallows coincident shells. Generally Dash should be used with the default value of these two commands, other options are provided primarily for compatibility with ACME.

If the `DISALLOW` option is selected then any time that coincident contact faces are found the code will abort with an error message indicating which elements were found to be coincident. The `DISALLOW` option should be used if you do not want any coincident shells to be considered in the analysis. The option operates essentially as a check on the mesh.

If the `IGNORE` option is selected, any contact faces attached to coincident shells or coincident shell/hexahedral boundaries are ignored for contact.

The default `BEST` and `TAPERED` options indicate that dash should use its standard shell lofting logic on coincident shells. For stacked shells this effectively means the shell stack has a contact thickness governed by the thickest shell in the stack. For coincident shell/hexahedral boundaries contact will be done on the shell faces which may be lofted somewhat away from the hexahedral face.

8.9.6 Search Options

```
BEGIN SEARCH OPTIONS [<string>name]
#
# Search tolerances
NORMAL TOLERANCE = <real>norm_tol
TANGENTIAL TOLERANCE = <real>tang_tol
CAPTURE TOLERANCE = <real>cap_tol
TENSION RELEASE = <real>ten_release
SLIP PENALTY = <real>slip_pen
END [SEARCH OPTIONS <string>name]
```

Implicit

Implicit

Implicit

Contact involves a search phase and an enforcement phase. The contact search algorithm used to detect interactions between contact surfaces is often the most computationally expensive part of an analysis. The user can exert some control over how the search phase is carried out via the `SEARCH OPTIONS` command block. By selecting different options in this command block, the user can make trade-offs between the accuracy of the search and computing time.

The `SEARCH OPTIONS` command block begins with the input line:

```
BEGIN SEARCH OPTIONS [<string>name]
```

and ends with:

```
END [SEARCH OPTIONS <string>name]
```

The `name` for the command block is optional.

Without a `SEARCH OPTIONS` command block, the default search with associated default search parameters is used for all contact pairs. If you want to override the default search method for all contact pairs, you should add a `SEARCH OPTIONS` command block. By adding a `SEARCH OPTIONS` command block, you establish a new set of global defaults for the search for all contact pairs.

The valid command lines within a `SEARCH OPTIONS` command block are described in Section 8.9.6 and Section 8.9.6.1. The values specified by these commands are applied by default to all interaction contact surfaces, unless overridden by a specific interaction definition.

8.9.6.1 Search Tolerances

```
NORMAL TOLERANCE = <real>norm_tol
TANGENTIAL TOLERANCE = <real>tang_tol
```

As indicated previously, the contact functionality in Sierra/SM uses a box defined around each face to locate nodes that may potentially contact the face. This box is defined by a tolerance normal to the face and another tolerance tangential to the face (see Figure 8.21). The code adds to these tolerances the maximum motion over a time step when identifying interactions. In the above command lines, the parameter `norm_tol` is the normal tolerance (defined on the `NORMAL TOLERANCE` command line) for the search box and the parameter `tang_tol` is the tangential tolerance (defined on the `TANGENTIAL TOLERANCE` command line) for the search box.

By default, Sierra/SM will automatically calculate normal and tangential tolerances roughly based on the characteristic length of a face multiplied by the value input by the search length scaling. The default search length scaling is 0.15. The automatic tolerances add the maximum motion over a time step just like the user defined tolerances. If you leave automatic search on and also specify normal and/or tangential tolerances with the `NORMAL TOLERANCE` and `TANGENTIAL TOLERANCE` command lines, the larger of the two (automatic or user specified) tolerances will be used. For example, suppose you specify a normal tolerance of 1.0×10^{-3} and the automatic tolerancing computes a normal tolerance of 1.05×10^{-3} . Then Sierra/SM will use a normal tolerance of 1.05×10^{-3} .

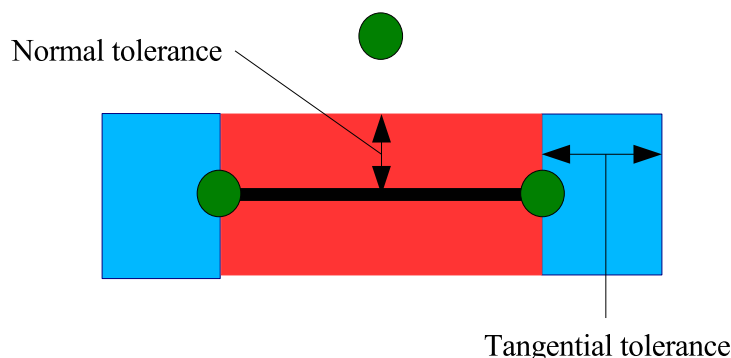


Figure 8.21: Illustration of normal and tangential tolerances

Both of these tolerances are absolute distances in the same units as the analysis. The proper tolerances are problem dependent. If a normal or tangential tolerance is specified in the `SEARCH OPTIONS` command block, they apply to all interactions. These default search tolerances can be overwritten for a specific interaction by specifying a value for the normal tolerance and/or tangential tolerance for that interaction inside the `INTERACTION` command block (see Section 8.6.2).

Normal and tangential contact tolerances are generally robustly automatically set and manual setting of tolerances should be avoided. If specific normal and tangential tolerances are set the tolerances used should not exceed roughly half of the minimum characteristic face size. Tolerances beyond that length may cause instability in the contact solution.



8.9.7 Enforcement Options

```
BEGIN ENFORCEMENT OPTIONS [<string>name]
  MOMENTUM BALANCE ITERATIONS = <integer>num_iterations(5)
  NUM GEOMETRY UPDATE ITERATIONS = <integer>num_updates(1)
  ACCUMULATED GAP REMOVAL OPTION =
```



```

REMOVE_ALL|REMOVE_DEFAULT| REMOVE_NONE (REMOVE_DEFAULT)
END [ENFORCEMENT OPTIONS <string>name]

```

Contact involves a search phase to create constraints and an enforcement phase to enforce those constraints. The user can exert some control over how the enforcement phase is carried out via the `ENFORCEMENT OPTIONS` command block. By selecting different options in this command block, the user can make trade-offs between solution accuracy and computing time. The `ENFORCEMENT OPTIONS` command block begins with the input line:

```

BEGIN ENFORCEMENT OPTIONS [<string>name]

```

and is terminated with the input line:

```

END [ENFORCEMENT OPTIONS <string>name]

```

The name for the command block, `name`, is optional

Only a single `ENFORCEMENT OPTIONS` command block is permitted within a `CONTACT DEFINITION` command block. Without an `ENFORCEMENT OPTIONS` command block, the default enforcement algorithm with associated default enforcement options is used for all contact pairs. If you want to override the defaults for enforcement for all contact pairs, you should add an `ENFORCEMENT OPTIONS` command block. By adding this command block, you establish a new set of global defaults for enforcement for all contact pairs.

The momentum balance algorithm for enforcement of the contact constraints uses an iterative process to ensure incremental momentum balance over a time step. Rather than making one pass to compute contact forces for node push-back, several passes are made to more accurately compute the normal contact force and, subsequently, the tangential (frictional) contact forces. The number of passes (iterations) is set by the value `num_iterations` in the `MOMENTUM BALANCE ITERATIONS` command line. The default value for the number of iterations is 5. This value is generally acceptable for satisfying the material interpenetration constraint. To get accurate results in a global sense in analyses that use friction, a value of 10 is more appropriate. To get accurate contact response at individual points in analyses with friction, a value of 20 or greater may be needed. Note that as the number of iterations increases, the expense of enforcement increases. Thus a user can balance execution speed and accuracy with this command line, though care must be taken to ensure that the appropriate level of accuracy is attained. This command line affects only the enforcement phase of the contact. A single search phase is used for contact detection, but the enforcement phase uses an iterative process.

The `NUM GEOMETRY UPDATE ITERATIONS` defines the number of geometry updates contact will take. When using geometry updates contact iterations will be redefined in the new predicted geometry increasing accuracy, but also significantly increasing cost.

The `ACCUMULATED GAP REMOVAL OPTION` controls how the explicit contact algorithm deals with either initial or accumulated gap in the frictionless and coulomb friction sliding interactions. Consider a mesh with initial overlap between contacting bodies. If the `OVERLAP REMOVAL` is not done then this initial overlap will be removed via standard contact forces. The `ACCUMULATED GAP REMOVAL OPTION` command will affect this case as follows:

- The `REMOVE_ALL` option will remove all this existing gap over a single time step. The acceleration and force required to do this removal over the single explicit step may be very high, and may add a lot of energy to the system. The remove all option will tend to prevent any overlap from ever occurring in the mesh, but can lead to spuriously high energy and contact stability problems in some cases. Note, the `REMOVE_ALL` option was the default interaction behavior before the 4.38 code release.
- The `REMOVE_NONE` will not remove any of the old gap. The contact constraint zero gap point will be defined with respect to the initial configuration. Thus the initial gap will not be removed and the initial overlap will not add energy. However, with the `REMOVE_NONE` option it is possible for small errors in enforcement to accumulate gap in the interactions over time, this can lead to contact interactions being lost and bodies passing through one another.
- The `REMOVE_DEFAULT` uses a heuristic weighting to remove some of the preexisting gap. The portion of preexisting gap removed will depend on the current gap and gap rate. Generally enough gap will be removed to ensure that accumulation of error will not cause loss of the interaction while the energy gain from preexisting gap removal is minimized. Preexisting gaps will be removed over several time steps. The default option provides a good balance of stability and contact accuracy and should be used for most circumstances.

Note, in addition to initial overlap preexisting gap can occur in many other circumstances including element death, particle creation, non-exact contact enforcement, and a variety of geometric ambiguities that can occur at corners. Thus selection of the gap removal option will have some, though generally minimal, effect on most runs.

8.10 Contact-Specific Outputs

Contact variables can be output to provide information about enforcement of contact interactions. Currently, information on only one interaction at each node is provided. If a node has more than one interaction, the last one in its internal interaction list is reported.

Nodal contact variables that can be output are listed in Table 9.8. Where applicable, names of the equivalent variables in JAS3D are given in parentheses at the end of the description. The variables can be output in history files or results files; see Chapter 9 for more information on outputting nodal variables. Note that currently the variables cannot be calculated at output time so the first time they are output a request is made to calculate them. This means that the first output step where they are to appear the data will be all zero. A work around for this is to have at least one output step in which these variables appear before their values are needed.

It is no longer required to specify `COMPUTE CONTACT VARIABLES = ON` in the input file. Now, Sierra/SMwill determine if a contact variables is being requested for output and compute that variable by default at every output time (unless it is an accumulated contact variable). If you would like to compute contact variables at every step (to be used in user output for example), specify `COMPUTE CONTACT VARIABLES = EVERY_STEP`

The contact variables that are accumulated every step and cannot only be computed at the output times are the following: `CONTACT_ACCUMULATED_SLIP`, `CONTACT_ACCUMULATED_SLIP_VECTOR`, `CONTACT_FRICTIONAL_ENERGY`, and `CONTACT_FRICTIONAL_ENERGY_DENSITY`.

8.10.1 Contact Debugging

```
BEGIN DEBUG
  VISUALIZE CONTACT FACETS = OFF|ON(OFF)
  OUTPUT CONTACT SURFACES AND CONSTRAINTS = OFF|ON(OFF)
  OUTPUT LEVEL = <integer>level(0)
  WRITE SALINAS MPCs TO <string>file AT TIME <real>time
END
```

Note: This option is available only for Dash contact.

The debug contact block enables additional information to be output by the code to help debug what may be going wrong with a contact problem.

The `OUTPUT LEVEL` command controls the amount of information written to the log file. At the default level of zero, only minimal information is included. At level 1 or greater, more information is printed to the log file such as the composition of the different contact surfaces and which friction model is being used for which surface pairing.

The `VISUALIZE CONTACT FACETS` command in the `DEBUG` command block can be used to plot the actual contact facets and lofted contact geometries that contact is being evaluated on. When this command is set to `ON`, extra element blocks and node blocks will be added to the output mesh to represent the contact geometries. Note that use of this option will significantly increase the size of the results file and moderately increase the run time. The naming convention used for these surfaces is to prepend the prefix `CSURF_` to the original contact surface name. For example, if a

contact surface is produced by skinning `block_1`, then the a side set will be created with the name `CSURF_block_1`. This is a Dash only option.

The `OUTPUT CONTACT SURFACES AND CONSTRAINTS` command block can be used to plot interactions into the mesh output file as sets of surfaces and nodes. Viewing the interactions can help debug contact problems. This feature only works with `node_face` contact interactions and only if the constraint can fit in the connectivity of a ten-noded tetrahedral element.

The `WRITE SALINAS MPCs` command can be used to write the contact constraints to a text file compatible with the Salinas input deck format. The constraints will be written to the specified file name `file` *once* at the first analysis time step after the specified target time `time`. All contact constraints (tied, glued, frictionless, frictional, etc.) are output in Salinas format as tied MPCs. The `WRITE SALINAS MPCs` command can be specified multiple times to output constraints at various times to various separate files.

8.11 Log File Contact Information

When contact is included in the analysis, pertinent information is written to the log file. Information on the name and size of each contact surface will be included in a table as shown:

----- DASH SURFACES -----					
Contact Entity	Num Faces	Num Lofted	Num Points	MinCharLen	MaxCharLen
pin_surf	0	0	287	0.0	0.012
block_1	994	0	0	0.01	0.01
block_2	0	24	0	0.02	0.03
surface_1000	4	0	0	0.50	0.50

The above table will have one entry for each defined contact surface. The name of the contact surface is in the “Contact Entity” column. “Num Faces” is the number of non-lofted faces (such as triangles or quadrilaterals) contained in the surface. “Num Lofted” is the number of lofted objects (such as shells, beams, or particle elements) contained in the surface. “Num Points” is the number of nodes in the contact surface that are not part of any of the faces or lofted elements. “MinCharLen” and “MaxCharLen” are the minimum and maximum characteristic length of any face or lofted object in the contact surface.

The log file will also include a representation of the interaction matrix, for example, of the form:

```

----- DASH INTERACTION WEIGHT MATRIX -----
0000
1H00
00HH
00H0

```

The topmost row and leftmost column of the interaction matrix correspond to the first entry in the “Contact Entity” list. The bottom most row and the rightmost column of the interaction matrix corresponds to the last entry in the “Contact Entity” list. The entry at row I, column J of the matrix describes how contact surfaces I and J interact.

A value of “1” indicates that contact surface I has interactions with contact surface J where surface I is the slave surface and surface J is the master surface. For node-face contact this means that the nodes of surface I contact the faces of surface J. For face-face contact this means that the faces of surface I interact with the faces of surface J and that the gap integral is performed on faces of surface I.

A value of “H” (short for half) indicates contact surface I interacts with contact surface J in symmetric contact. For node-face contact this means that the nodes of surface I contact the faces of surface J and also the nodes of surface J contact the faces of surface I. For face-face contact this means the faces of surface I contact the faces of surface J and the gap integral is performed simultaneously on both sides of the interface.

A value of “0” in both the IJ and JI position of the matrix indicates that surfaces I and J will not interact via contact.

Note that a surface can only interact with itself (the diagonal entries of the matrix) through self contact. If entry IJ of the matrix is “1” entry JI must be “0”. If entry IJ of the matrix is “H” than entry JI must also be “H”.

8.12 Examples

This section has several example problems. The geometric configuration for each problem is presented together with a brief description and the appropriate command lines.

8.12.0.1 Example 1

The first example problem has two blocks that come into contact due to initial velocity conditions. Block 1 has an initial velocity equal to v_1 , and block 2 has an initial velocity equal to v_2 . The geometric configuration for this problem is shown in Figure 8.22.

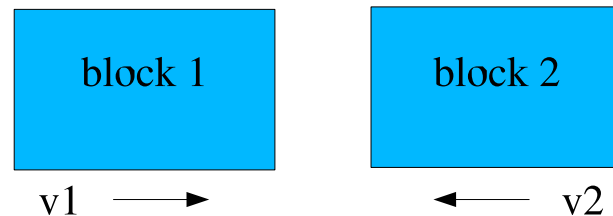


Figure 8.22: Problem with two blocks coming into contact

The simplest input for this problem will be named `EXAMPLE1` and is shown as follows:

```
BEGIN CONTACT DEFINITION EXAMPLE1
#
# define contact surfaces
SKIN ALL BLOCKS = ON

# set interactions
BEGIN INTERACTION DEFAULTS
  GENERAL CONTACT = ON
END

END
```

In this example, the `SKIN ALL BLOCKS` command line with parameter set to `ON` will create a surface named `block_1` from the exterior skin of `block_1` and a surface named `block_2` from the exterior skin of `block_2`. The `GENERAL CONTACT = ON` command defines that `block_1` can contact `block_2`. As no friction model was specified, the default frictionless model is used for this interaction.

Now consider the same problem (two blocks coming into contact) in which it is desired that the two blocks interact with a coulomb friction model. The input for this variation of the two block problem will be named `EXAMPLE1A` and is shown as follows:

```
BEGIN CONTACT DEFINITION EXAMPLE1A
#
```

```

# define contact surfaces
SKIN ALL BLOCKS = ON

# friction model definition
BEGIN CONSTANT FRICTION MODEL rough
    FRICTION COEFFICIENT = 0.5
END

# set interactions
BEGIN INTERACTION EX1A
    SURFACES = block_1 block_2
    FRICTION MODEL = rough
END
END

```

As was the case for the `EXAMPLE1` command block, the `SKIN ALL BLOCKS` command line with its parameter set to `ON` will create a surface named `block_1` from the skin of `block_1` and a surface named `block_2` from the skinning of `block_2`.

In order to have frictional contact between the two blocks, a constant Coulomb friction model is specified with a `CONSTANT FRICTION MODEL` command block. We name this model `ROUGH` and specify the coefficient of friction as `0.5`.

In the `INTERACTION` command block, the friction model `ROUGH` is selected on the `FRICTION MODEL` command line. This model is then applied to the interaction of surfaces `block_1` and `block_2`.

8.12.0.2 Example 2

A second more complex example problem has three blocks that come into contact due to initial velocity conditions. Block 1 is a solid element brick block with initial velocity equal to v_1 . Block 3 is a solid element brick block with initial velocity v_3 . Block 2 is a shell element block that has zero initial velocity. There is a side set (Surface 3) on the top faces of block 3. The geometric configuration for this problem is shown in Figure 8.23. The contact input for this three-block problem is called `EXAMPLE2` and is shown as follows:

```

BEGIN CONTACT DEFINITION EXAMPLE2
#
# define contact surfaces
CONTACT SURFACE b1 CONTAINS block_1
CONTACT SURFACE b2 CONTAINS block_2
CONTACT SURFACE s3 CONTAINS surface_3

BEGIN CONSTANT FRICTION MODEL rough
    FRICTION COEFFICIENT = 0.5
END
#
# set interactions

```

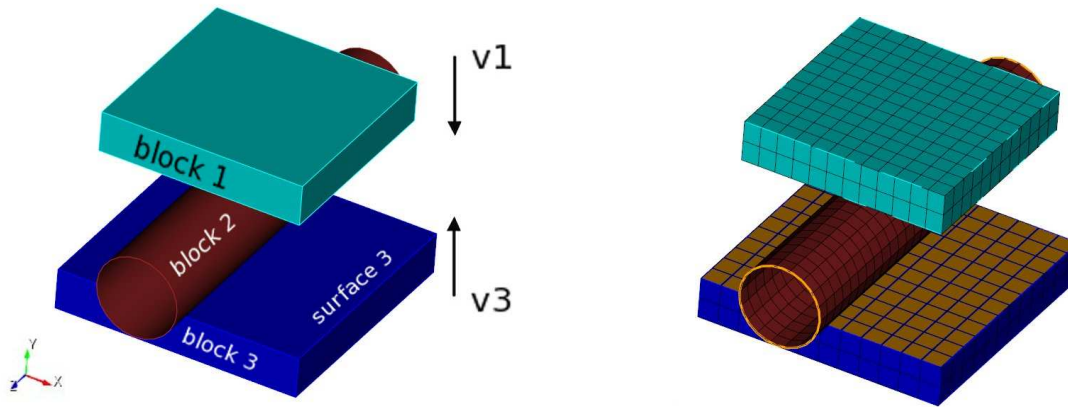


Figure 8.23: Problem with three blocks coming into contact.

```

BEGIN INTERACTION DEFAULTS
  FRICTION MODEL = rough
  GENERAL CONTACT = ON
  SELF CONTACT = ON
END

# set specific interaction
BEGIN INTERACTION
  SURFACES = b2 s3
  FRICTION MODEL = FRICTIONLESS
END

END

```

In the `EXAMPLE2` command block, three surfaces have been defined. The first surface, `b1`, is obtained by skinning `block_1`. The second surface, `b2` is obtained by skinning `block_2`. The third surface, `s3`, is the user-defined side set `surface_3`.

In the `INTERACTION DEFAULTS` command block, we select the friction model `ROUGH` on the `FRICTION MODEL` command line. Both `GENERAL CONTACT` and `SELF CONTACT` are set to `ON`. For this problem the hollow cylinder, `block_2`, will undergo self-contact. Setting `GENERAL CONTACT` to `ON` will enforce contact between `b1` and `b2`, `b2` and `s3`, and `b1` and `s3`. Setting `SELF CONTACT` to `ON` will enable self-contact for all three of the surfaces.

For this particular example, we want to override some of the default setting for surface interactions which were previously set by the `INTERACTION DEFAULTS` command block. The `INTERACTION` command block shown in `EXAMPLE2` overrides these defaults and instead specifies surfaces `b2` and `s3` to have a frictionless interaction.

In Figure 8.24, the bodies are shown after deformation and contact. The second image shows a cross section cut through the center. Due to the lofting of the shell surface there is a small gap between the shell elements themselves and the compressing blocks, which is expected. The actual contact geometry is shown in Figure 8.25. Note that as Block 1 is skinned, the entire exterior

surface of block 1 is included in contact surface b1. The lofted geometry of the shell blocks is seen in this figure, this lofted geometry is both contacting with the blocks and experiencing self contact. Contact surface s3 contains only the top sideset on block 3, not the entire skin.

Time = 0.000500

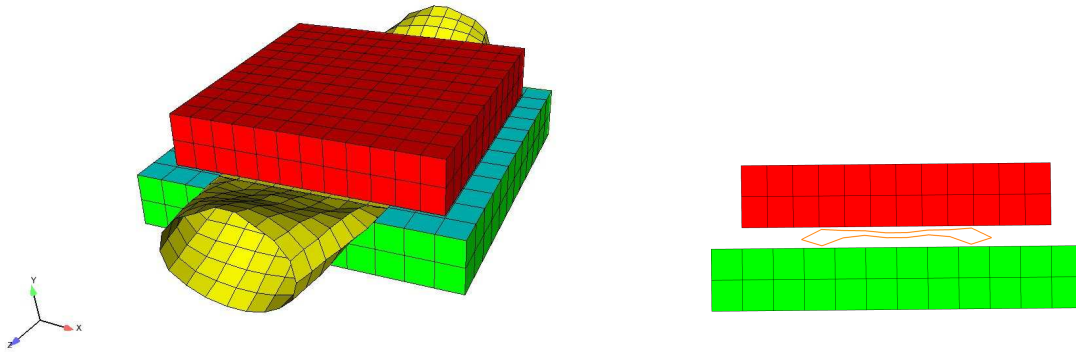


Figure 8.24: State of bodies after contact and deformation.

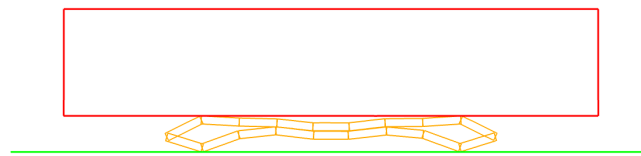


Figure 8.25: Deformed contact geometry.

8.13 Common Contact Algorithmic Issues

This section describes some of the common problem issues that may be encountered when setting up contact.

Contact constraints are one type of constraint among many possible types of constraints. Other constraint types include MPCs 7.14, kinematic constraints 7, and welds 7.14. Contact constraints can potentially conflict with these other constraint types causing unexpected behavior. Refer to Appendix D for information on how conflicting constraints are handled.

Tied contact that occurs over a substantial distance is not guaranteed to preserve energy or angular momentum at the interface. The issue becomes more problematic as the distance between tied objects becomes a substantial fraction of the contact face size. It is recommended that default search tolerances are used so that only objects in close proximity are tied.

Lofted objects that are large compared to the element size are problematic. For example, lofting of shells that are thicker than they are wide or beams with a large radius compared to their element length.

The speed of contact is based primarily on the number of nodes and faces in the contact surfaces and, to a lesser extent, on the number of interactions specified. Choosing excessively large search tolerances will greatly increase the run time required by contact.

Contact performs best when objects in contact are on the same processor. It is strongly recommended to use a proximity based mesh decomposition method such as RCB or inertial for contact problems. If large motions occur during the problem solution, dynamic mesh rebalancing may be needed to maintain good performance, see Section 6.8.

The glued friction model is enforced by a velocity constraint. Over a very large number of steps, small integration errors can build up and potentially allow for some slip at the glued interface.

Large local mesh motions may cause inversion of the contact interface. It is recommended that shape metric based element death or other mechanisms be used to prevent the large errors that can occur when computing contact on tangled surfaces. Ensuring the nodal jacobian ratio of all elements is greater than zero will usually ensure the integrity of the contact surface definition.

Even when using frictionless interaction laws, contact interfaces will tend to be energy absorbers. This is due to contact interfaces behaving plastically rather than storing impact energy.

Initial overlap of contacting bodies may cause major instabilities in explicit contact runs. The remove initial overlap capability can be used to remove small overlap. Large overlaps must be removed by modifying the underlying geometry or running a preload step.

The implicit and explicit contact solvers behave subtly different. If an implicit static preload is used and that state handed off to an explicit analysis, there will usually be a small equilibrium imbalance that needs to ring down after the transfer.

8.14 ACME Specific Commands

The Dash contact search is used by default and is the recommended methodology for all analyses. The ACME search is still maintained for backwards compatibility. this section lists ACME specific contact commands and behavior. The ACME search may be deprecated in the future leaving Dash as the only supported option.

8.14.1 ACME Specific Friction Models

The below friction models are available when using ACME search in an explicit dynamics analysis.



8.14.1.1 Spring Weld Model

```
BEGIN SPRING WELD MODEL <string>name
  NORMAL DISPLACEMENT FUNCTION = <string>func_name
  NORMAL DISPLACEMENT SCALE FACTOR = <real>scale_factor(1.0)
  TANGENTIAL DISPLACEMENT FUNCTION = <string>func_name
  TANGENTIAL DISPLACEMENT SCALE FACTOR =
    <real>scale_factor(1.0)
  FAILURE ENVELOPE EXPONENT = <real>exponent(2.0)
  FAILURE DECAY CYCLES = <integer> num_cycles(1)
  FAILED MODEL = <string>failed_model_name|FRICTIONLESS
    (FRICTIONLESS)
END [SPRING WELD MODEL <string>name]
```

The `SPRING WELD MODEL` command block defines a contact friction model that, when applied between two contact surfaces, connects a slave node to the nearest point of a corresponding master face with a spring. The spring behavior is defined by a force-displacement curve in the normal and tangential directions. If the motion of the problem generates displacement between the slave node and its corresponding master face and this motion is in purely the normal or tangential direction, the spring will fail once it passes the maximum displacement in the normal and tangential force-displacement curves, respectively. For displacements that include both normal and tangential components, the spring fails according to a failure criterion defined as the sum of the ratios of the normal and tangential components to their maximum values, raised to a power. If the criterion is greater than 1.0, the spring fails. Once the spring fails, its contact forces reduce over a number of load steps, and the contact evaluation reverts to another user-specified friction model (or frictionless contact if not specified).

In the above command block:

- The string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions.
- The normal force-displacement curve is specified by the `NORMAL DISPLACEMENT FUNCTION` command line, where the string `func_name` is the name of a function defined in a `FUNCTION` command line in the `SIERRA` scope. This function can be scaled by the real value `scale_factor` in the `NORMAL DISPLACEMENT SCALE FACTOR` command line; the default for this factor is 1.0
- The tangential force-displacement curve is specified by the `TANGENTIAL DISPLACEMENT`

FUNCTION command line, where the string `func_name` is the name of a function defined in a DEFINITION FOR FUNCTION command line in the SIERRA scope. This function can be scaled by the real value `scale_factor` in the TANGENTIAL DISPLACEMENT SCALE FACTOR command line; the default for this factor is 1.0.

- The real value `exponent` in the FAILURE ENVELOPE EXPONENT command line specifies how normal and tangential failure criteria may be combined to yield failure of the weld, as described above. The default value for this exponent is 2.0.
- The FAILURE DECAY CYCLES command line describes how many cycles to ramp down the load in the spring weld after it fails through the integer value `num_cycles`. The default value for the number of decay cycles is 1.
- When the spring weld breaks, the friction model that contact reverts to when evaluating future node-face interactions between the surfaces is identified in the FAILED MODEL command line with the string `failed_model_name`. The friction model listed in this command must have been previously defined in the input file. The default value for the model used after failure is the frictionless model.

The SPRING WELD MODEL command block is similar to the Sierra/SM SPOT WELD command block, but permits greater flexibility in specifying a different friction model to be applied after failure.



8.14.1.2 Surface Weld Model

```
BEGIN SURFACE WELD MODEL <string>name
  NORMAL CAPACITY = <real>normal_cap
  TANGENTIAL CAPACITY = <real>tangential_cap
  FAILURE DECAY CYCLES = <integer>num_cycles(1)
  FAILED MODEL = <string>failed_model_name|FRICTIONLESS
  (FRICTIONLESS)
END [SURFACE WELD MODEL <string>name]
```

The SURFACE WELD MODEL command block defines a contact friction model that behaves identically to the TIED MODEL until a maximum force between the node and face of an interaction is reached in the normal direction or the tangential direction. Once this maximum force is reached, the tied contact “fails” and the friction model switches to a different friction model, as specified by the user.

In the above command block, the string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions. The maximum allowed force in the normal direction is specified by the real value `normal_cap` in the NORMAL CAPACITY command line. The maximum allowed force in the tangential direction is specified by the real value `tangential_cap` in the TANGENTIAL CAPACITY command line. There are no defaults for these values. The surface weld will break when either the specified normal or tangential capacity is reached. Once the model fails, the applied forces decrease to zero over a number of time steps defined through the integer value `num_cycles` in the FAILURE DECAY CYCLES command line. The default for `num_cycles` is 1. The friction model that should be used after the weld fails is identified in the FAILED MODEL command line with the string `failed_model_name`. The friction model designated in the FAILED MODEL command line must be defined within the CONTACT DEFINITION command



block. The default model after failure is the frictionless contact model.

8.14.1.3 Area Weld Model

```
BEGIN AREA WELD MODEL <string>name
  NORMAL CAPACITY = <real>normal_cap
  TANGENTIAL CAPACITY = <real>tangential_cap
  FAILURE DECAY CYCLES = <integer> num_cycles(1)
  FAILED MODEL = <string>failed_model_name|FRICTIONLESS
    (FRICTIONLESS)
END [AREA WELD MODEL <string>name]
```

The `AREA WELD MODEL` command block defines a contact friction model that behaves identically to the `TIED MODEL` until a maximum traction between a node and face in an interaction is reached in the normal direction or the tangential direction. Once this maximum traction is reached, the tied contact “fails” and the friction model switches to a different friction model, as specified by the user. This model is identical to the `SURFACE WELD MODEL` command block, except that tractions are used instead of forces.

In the above command block, the string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions. The maximum allowed traction in the normal direction is specified by the real value `normal_cap` in the `NORMAL CAPACITY` command line. The maximum allowed traction in the tangential direction is specified by the real value `tangential_cap` in the `TANGENTIAL CAPACITY` command line. There are no defaults for these values. The area weld will break when either the specified normal or tangential capacity is reached. Once the model fails, the applied tractions decrease to zero over a number of time steps defined through the integer value `num_cycles` in the `FAILURE DECAY CYCLES` command line. The default for `num_cycles` is 1. The friction model that should be used after the weld fails is identified in the `FAILED MODEL` command line with the string `failed_model_name`. The friction model designated in the `FAILED MODEL` command line must be defined within the `CONTACT DEFINITION` command block. The default model after failure is the frictionless contact model.



8.14.1.4 Adhesion Model

```
BEGIN ADHESION MODEL <string>name
  ADHESION FUNCTION = <string>func_name
  ADHESION SCALE FACTOR = <real>scale_factor(1.0)
END [ADHESION MODEL <string>name]
```

The `ADHESION MODEL` command block defines a friction model that behaves like frictionless contact when two surfaces are in contact, but computes an additional force between the surfaces when they are not touching. The value of the additional force is given by a user-specified function of force versus distance, where the distance is the distance between a node and the closest point on the opposing surface.

In the above command block, the string `name` is a user-selected name for this friction model that is used to identify this model in the interaction definitions. The force between surfaces that are not

touching is given by the `ADHESION FUNCTION` command line, where the string `func_name` is the name of a function defined in a `FUNCTION` command block in the SIERRA scope. The values of this function are expected to be non-negative. The function can be scaled by the real value `scale_factor` in the `ADHESION SCALE FACTOR` command line; the default for this factor is 1.0. Because contact forces are typically only given to node-face interactions if they touching, the contact search requires appropriate tolerances when this model is used. The normal and tangential tolerances specified in the interaction definitions should be set to the maximum distance at which the adhesion model should be applying force. However, setting this distance to be large may cause excessive numbers of interactions to be identified in the search phase, causing the contact processing to be extremely slow and/or generate erroneous interactions.



Explicit Only

8.14.1.5 Junction Model

```
BEGIN JUNCTION MODEL <string>name
  NORMAL TRACTION FUNCTION = <string>func_name
  NORMAL TRACTION SCALE FACTOR = <real>scale_factor(1.0)
  TANGENTIAL TRACTION FUNCTION = <string>func_name
  TANGENTIAL TRACTION SCALE FACTOR = <real>scale_factor(1.0)
  NORMAL CUTOFF DISTANCE FOR TANGENTIAL TRACTION = <real>distance
END [JUNCTION MODEL <string>name]
```

Note: This model is available only in explicit dynamics with ACME contact

The `JUNCTION MODEL` command block defines a model that prevents the interpenetration of contact surfaces and that also provides normal and tangential tractions to a node-face interaction when the surfaces are not touching. The normal tractions are defined as a function of the normal distance between the node and face of an interaction, while the tangential traction is given as a function of the relative tangential velocity. The tractions are defined by user-specified functions, and the tangential tractions from this model drop to zero once the normal distance between the node and the face exceeds a critical value. This friction model provides a simple way to model threaded connections, though the `THREADED MODEL` defined in Section 8.14.1.6 has more flexibility.

In the above command block, the string `name` is a user-selected name for this friction model that is used to identify this model in the interaction blocks. The normal traction curve is specified by the `NORMAL TRACTION FUNCTION` command line, where the string `func_name` is the name of a function defined in a `DEFINITION FOR FUNCTION` command block in the SIERRA scope. This function defines a relation between the traction and the distance between the node and the face in the normal direction. This function can be scaled by the real value `scale_factor` in the `NORMAL TRACTION SCALE FACTOR` command line; the default for this factor is 1.0. Similarly, the tangential traction curve is specified by the `TANGENTIAL TRACTION FUNCTION` command line, where the string `func_name` is the name of a function defined in a `FUNCTION` command block in the SIERRA scope. This function defines a relation between the traction and the relative velocity of the node and face in the tangential direction. This function can be scaled by the real value `scale_factor` in the `TANGENTIAL TRACTION SCALE FACTOR` command line; the default for this factor is 1.0. Once the normal distance between a node and a face using this model reaches a critical distance, the tangential traction drops to zero; this distance is specified with the real value `distance` in the `NORMAL CUTOFF DISTANCE FOR TANGENTIAL TRACTION` command line.



8.14.1.6 Threaded Model

```
BEGIN THREADED MODEL <string>name
  NORMAL TRACTION FUNCTION = <string>func_name
  NORMAL TRACTION SCALE FACTOR = <real>scale_factor(1.0)
  TANGENTIAL TRACTION FUNCTION = <string>func_name
  TANGENTIAL TRACTION SCALE FACTOR = <real>scale_factor(1.0)
  TANGENTIAL TRACTION GAP FUNCTION = <string>func_name
  TANGENTIAL TRACTION GAP SCALE FACTOR = <real>scale_factor(1.0)
  NORMAL CAPACITY = <real>normal_cap
  TANGENTIAL CAPACITY = <real>tangential_cap
  FAILURE ENVELOPE EXPONENT = <real>exponent(2.0)
  FAILURE DECAY CYCLES = <integer>num_cycles(1)
  FAILED MODEL = <string>failed_model_name|FRICTIONLESS
    (FRICTIONLESS)
END [THREADED MODEL <string>name]
```

The `THREADED MODEL` command block defines a friction model that is designed to mimic a threaded interface. This model prevents interpenetration of contact surfaces, and also supplies additional tractions when the surfaces are not touching. Tensile tractions in the normal direction are given by a user-specified function of force versus distance between the node and face. Tensile tractions in the tangential direction are computed as the product of a traction tangential-displacement curve and a scaling curve that is a function of the normal displacement. Maximum normal and tangential tractions are input such that the model “fails” at a node-face interaction once they are reached. For interactions that include both normal and tangential displacements, the model failure is defined according to a failure criterion defined as the sum of the ratios of the normal and tangential traction components to their maximum capacity values, raised to a power. After failure, interactions shift to a different user-specified friction model.

In the above command block:

- The string `name` is a user-selected name for this friction model that is used to identify the model in the interaction definitions.
- The traction-displacement relation in the normal direction is specified by the `NORMAL TRACTION FUNCTION` command line, where the string `func_name` is the name of a function defined in a `FUNCTION` command block in the `SIERRA` scope. This function can be scaled by the real value `scale_factor` in the `NORMAL TRACTION SCALE FACTOR` command line; the default for this factor is 1.0.
- The traction-displacement relation in the tangential direction is specified by two curves. The traction-displacement relation in the tangential direction when there is no displacement in the normal direction is defined by the `TANGENTIAL TRACTION FUNCTION` command line, where the string `func_name` is the name of a function defined in a `FUNCTION` command block in the `SIERRA` scope. This function can be scaled by the real value `scale_factor` in the `TANGENTIAL TRACTION SCALE FACTOR` command line; the default for this factor is 1.0. When the distance in the normal direction is greater than zero, the tangential traction is scaled by a function specified in the `TANGENTIAL TRACTION GAP FUNCTION` command line, where the string `func_name` is the name of a function defined in a `FUNCTION` command

block in the SIERRA scope. This function defines a scaling factor as a function of the normal displacement. The function can be scaled by the real value `scale_factor` in the TANGENTIAL TRACTION GAP SCALE FACTOR command line; the default for this factor is 1.0.

- The threaded model “fails” once the normal and tangential tractions reach a critical capacity value. The normal capacity is specified by the real value `normal_cap` in the NORMAL CAPACITY command line. The tangential capacity is specified by the real value `tangential_cap` in the TANGENTIAL CAPACITY command line. There are no default values for these parameters. These capacities are defined for pure normal or tangential displacements. In cases where there is a combination of tangential and normal displacements, a failure curve is used to determine the combination of tangential and normal tractions that determines model failure. This curve is defined as the sum of the ratios of the normal and tangential traction components to their maximum capacity values, raised to a power. The power in the function is defined by the real value `exponent` in the FAILURE ENVELOPE EXPONENT command line. The default value of the exponent is 2.0. Once the model fails, the applied tractions decrease to zero over a number of time steps defined through the integer value `num_cycles` in the FAILURE DECAY CYCLES command line. The default for `num_cycles` is 1. When the model exceeds the designated capacity, the contact surfaces using this model switch to a different friction model as identified in the FAILED MODEL command line with the string `failed_model_name`. The friction model designated in the FAILED MODEL command line must be defined within the CONTACT DEFINITION command block. The default model is the frictionless model.



Explicit Only

8.14.1.7 PV_Dependent Model

```
BEGIN PV_DEPENDENT MODEL <string>name
  STATIC COEFFICIENT = <real>stat_coeff
  DYNAMIC COEFFICIENT = <real>dyn_coeff
  VELOCITY DECAY = <real>vel_decay
  REFERENCE PRESSURE = <real>p_ref
  OFFSET PRESSURE = <real>p_off
  PRESSURE EXPONENT = <real>p_exp
END [PV_DEPENDENT MODEL <string> name]
```

The PV_DEPENDENT MODEL command block defines a friction model similar to a coulomb friction model, but which provides a frictional response that is dependent on the pressure and the velocity. The pressure-dependent portion of the model behaves similarly to the constant friction model except that the tangential traction is given by

$$\left[\frac{p + p_{\text{off}}}{p_{\text{ref}}} \right]^{p_{\text{exp}}} . \quad (8.3)$$

The velocity-dependent part is given by

$$(\text{stat\_coeff} - \text{dyn\_coeff}) e^{(-\text{vel\_decay}||v||)} + \text{dyn\_coeff}. \quad (8.4)$$

The `PV_DEPENDENT MODEL` command block multiplies the pressure and velocity effects together.

In the above command block:

- The string `name` is a name assigned to this friction model that is used to identify the model in the interaction definitions.
- The real value `p_ref` in the pressure-dependent part given in Equation (8.3) is specified with the `REFERENCE PRESSURE` command line.
- The real value `p_off` in the pressure-dependent part given in Equation (8.3) is specified with the `OFFSET PRESSURE` command line.
- The real value `p_exp` in the pressure-dependent part given in Equation (8.3) is specified with the `PRESSURE EXPONENT` command line.
- The real value `stat_coeff` in the velocity-dependent part given in Equation (8.4) is specified with the `STATIC COEFFICIENT` command line.
- The real value `dyn_coeff` in the velocity-dependent part given in Equation (8.4) is specified with the `DYNAMIC COEFFICIENT` command line.
- The real value `vel_decay` in the velocity-dependent part given in Equation (8.4) is specified with the `VELOCITY DECAY` command line.



8.14.1.8 User Subroutine Friction Models

```
BEGIN USER SUBROUTINE MODEL <string>name
  INITIALIZE MODEL SUBROUTINE = <string>init_model_name
  INITIALIZE TIME STEP SUBROUTINE = <string>init_ts_name
  INITIALIZE NODE STATE DATA SUBROUTINE =
    <string>init_node_data_name
  LIMIT FORCE SUBROUTINE = <string>limit_force_name
  ACTIVE SUBROUTINE = <string>active_name
  INTERACTION TYPE SUBROUTINE = <string>interaction_name
  NUM INTEGER VALS = <int>num_int_vals
  INTEGER VALS = <int>int_vals
  NUM REAL VALS = <int>num_real_vals
  REAL VALS = <real>real_val
  NUM HISTORY VALS = <int>num_hist
  USER FUNCTION <int>id =
    <string>name SCALED BY <real>scale
END [USER SUBROUTINE MODEL <string>name]
```

The `USER SUBROUTINE MODEL` command blocks permit contact to use a user subroutine to define a friction model between surfaces. This capability is in a test phase at this time; please contact a Sierra/SM developer for more information.

In this command block:

- The string `name` is a user-specified name that is used to identify this model in the interaction definitions.
- The command line `INITIALIZE MODEL SUBROUTINE` specifies a user subroutine to initialize the friction model. The name of the subroutine is given by `init_model_name`.
- The command line `INITIALIZE TIME STEP SUBROUTINE` specifies a user subroutine to

initialize the time step. The name of the subroutine is given by `init_ts_name`.

- The command line `INITIALIZE NODE STATE DATA SUBROUTINE` specifies a user subroutine to initialize the node state data. The name of the subroutine is given by `init_node_data_name`.
- The command line `LIMIT FORCE SUBROUTINE` specifies a user subroutine to provide the limit force for the friction model. The name of the subroutine is given by `limit_force_name`.
- The command line `ACTIVE SUBROUTINE` specifies a user subroutine to compute forces for an active node-face interaction. The name of the subroutine is given by `active_name`.
- The command line `INTERACTION TYPE SUBROUTINE` specifies a user subroutine to define the interaction type. The name of the subroutine is given by `interaction_name`.

For documentation of the remaining commands see the ACME manual.

8.14.2 Kinematic Partition

Kinematic partitioning is used only for explicit contact analysis. When computing two way contact between surface pairs ACME explicit enforcement uses a concept of kinematic partitioning to speed the convergence of the contact enforcement iterations.

The interaction defaults command block may contain the line

```
AUTOMATIC KINEMATIC PARTITION = <string>ON|OFF (OFF)
```

The interaction command block may contain the lines

```
AUTOMATIC KINEMATIC PARTITION = <string>ON|OFF (OFF)  
KINEMATIC PARTITION = <real>kin_part (0.5)
```

If the `AUTOMATIC KINEMATIC PARTITION` command line is used, Sierra/SM will automatically compute the kinematic partition factors for pairs of surfaces. The automatic kinematic partitions are computed from the impedance of each surface based on nodal average density and wave speed. Automatic computation of kinematic partition factors provides the best approach to exact enforcement of symmetric contact of opposing surfaces provided that these surfaces have the same mesh resolution. If the mesh resolution is disparate, you can specify the coarser meshed body as master, but generally it is better to use more contact iterations to deal with this case of a fine mesh contacting a coarse mesh.

For the interaction of any two surfaces, the sum of the partition factors for the surfaces must be 1.0. This is automatically taken care of when the `AUTOMATIC KINEMATIC PARTITION` command line is used. The default value for kinematic partition factors for all surfaces is 0.5.

The `AUTOMATIC KINEMATIC PARTITION` command line can be used to set the kinematic partitions for all interactions or to set the kinematic partitions for specific interactions. Thus the command line can appear in two different scopes:

1. The command line can be used within the `INTERACTION DEFAULTS` command block. In this case, all contact surface interactions defined in the command block will use the automatic

kinematic partitioning scheme by default. This will override the default case that assigns a kinematic partition factor of 0.5 to all surfaces. For particular interactions, it is possible to override the use of the automatic kinematic partition factors by specifying kinematic partition values (with the `KINEMATIC PARTITION` command line) within the `INTERACTION` command blocks for those interactions.

2. The command line can be used inside an `INTERACTION` command block. If the automatic partitioning command line appears inside an `INTERACTION` command block, the kinematic partition factors for that particular interaction will be calculated by the automatic kinematic partition scheme.

Automatic kinematic partitioning is not currently operational for shell elements. If it is enabled, the kinematic partitioning factor is set to 0.5 for all contact interactions involving shell elements.

To provide accurate contact evaluation, Sierra/SM typically computes two-way contact between two surfaces, where interactions are defined between the nodes of the first surface and the faces of the second surface, and also between the nodes of the second surface and the faces of the first surface. If the two surfaces have penetrated each other by a distance δ , then each of the contact evaluations will compute forces to move the surface a distance δ , so that the total resulting displacement would be 2δ if both sets of contact computations were fully applied. The `KINEMATIC PARTITION` command line defines a kinematic partition scaling factor, `kin_part`, for the two contact computations so that the total contact motion is correct. The kinematic partition factor, `kin_part`, is a value between 0.0 and 1.0. The factor scales the relative motion of the first surface, where `kin_part` = 0.0 means the first surface will move none of δ , while `kin_part` = 1.0 means the surface moves all of δ . The second surface moves the portion of δ that remains after the motion of the first surface (i.e., δ for second surface = $1.0 - \text{kin\_part}$). For instance, if `kin_part` is 0.2, the first surface will move 20% of the penetration distance (0.2δ in our example), and the second surface would move the remaining 80% (0.8δ in our example). The default value is 0.5, so that each surface would move half of the penetration distance. If `kin_part` is 0.0, the first surface does not move at all, and the second surface moves the full distance. This is exactly equivalent to a one-way master-slave contact definition, where the first surface is the master and the second is the slave. If `kin_part` is 1.0, the second surface is the master, and the first surface is the slave. Figure 8.26 illustrates how the kinematic partition factor varies from 0.0 to 1.0, with the specific example of `kin_part` being set to 0.2.

The capability provided by the `KINEMATIC PARTITION` command line is important in cases where contact occurs between two materials of disparate stiffness. Physically, we would expect a material with a higher stiffness to have more of an effect in determining the position of the contact surface than a more compliant material. In this case, we want the softer material to move more of the distance, and thus it should have a higher kinematic partition factor. The appropriate kinematic partition factor can be determined in closed form; see the ACME contact library reference [1] for more information. Alternately, the `AUTOMATIC KINEMATIC PARTITION` capability can automatically calculate the proper kinematic partition based on the stiffness of the materials.

Another case where the kinematic partition factor has traditionally been used is when meshes with dissimilar resolutions contact each other. If an interaction is defined with a fine mesh as the master surface and a coarse mesh as a slave surface, the contact algorithms will permit nodes on the

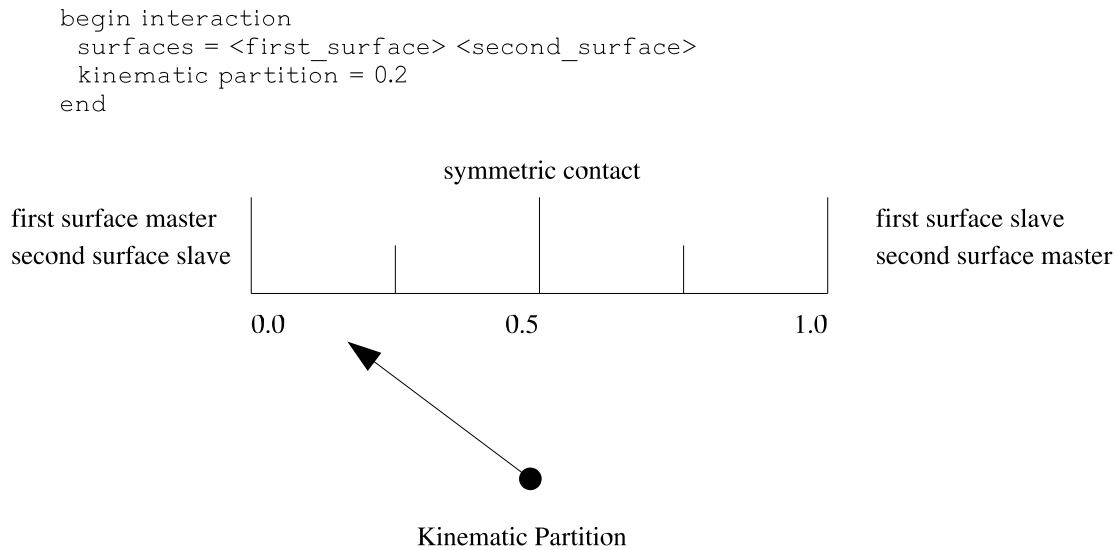


Figure 8.26: Illustration of kinematic partition values

master surface to penetrate the slave surface. In these cases, such problems can be alleviated by making the coarse mesh the master surface. However, the iterative approach implemented in the enforcement can also take care of this problem and is advised. If few iterations are chosen, an appropriate kinematic partition factor may be needed to prevent unintentional penetration due to mesh discretization.

For self-contact, the kinematic partition factor should be 0.5.

A kinematic partition factor cannot be defined for interactions that use the pure master-slave syntax (see Section 8.6.2.1).

In general, it is best to use the automatic kinematic partition option to properly compute the kinematic partition for a pair of surfaces. However, in a few cases, master-slave interactions are preferred. These cases consist of (1) interaction between a kinematically rigid surface and a deformable body, where the kinematically rigid surface should be the master surface; and (2) contact between shells and solids, where the solid should be the master surface.

8.14.3 ACME automated search tolerances

The methodology to compute default search tolerances can be set with the following commands in the SEARCH OPTIONS command blocks.

```
SEARCH TOLERANCE = AUTOMATIC|USER_DEFINED|DASH_AUTOMATIC(AUTOMATIC)
FACE MULTIPLIER = <real>face_multiplier(0.1)
```

By default both Dash and ACME contact searches define automated search tolerances. The search tolerance algorithms used by Dash and ACME contact search differ somewhat. It is possible to have

ACME use the Dash tolerances (which are often more robust) with use of the `DASH_AUTOMATIC` option to the `SEARCH TOLERANCE` command. If the `USER_DEFINED` option is used with the `SEARCH TOLERANCE` command ACME will use no automated search tolerances at all and rely only on the search tolerance values given in the input deck. When the `USER_DEFINED` option is used the normal and tangential tolerances must be specified. If these tolerances are not specified, code execution will be terminated with an error. The default `AUTOMATIC` search tolerance method computes automatic search tolerances based on the minimum characteristic length of any faces in the contact surface multiplied by the `face_multiplier`.

The face size multiplier used by the ACME `AUTOMATIC` tolerancing method can be set for all interactions by use of the `FACE MULTIPLIER` command in `SEARCH OPTIONS` command block. The `FACE MULTIPLIER` command can also be used in individual `INTERACTION` command blocks.

The automatic tolerances add the maximum motion over a time step just like the user defined tolerances. If you leave automatic search on and also specify normal and/or tangential tolerances with the `NORMAL TOLERANCE` and `TANGENTIAL TOLERANCE` command lines, the larger of the two (automatic or user specified) tolerances will be used. For example, suppose you specify a normal tolerance of 1.0×10^{-3} and the automatic tolerancing computes a normal tolerance of 1.05×10^{-3} . Then Sierra/SM will use a normal tolerance of 1.05×10^{-3} .



8.14.4 Frictional behavior without Friction Model

ACME may define coulomb frictional behavior without explicitly defining a coulomb friction model with use of the following command in an `INTERACTION` command block.

```

FRICION COEFFICIENT = <real>coeff
FRICION COEFFICIENT FUNCTION = <string>coeff_func

```

The `FRICION COEFFICIENT` command line allows setting a friction coefficient irrespective of the friction model used. The command line `FRICION COEFFICIENT FUNCTION` provides a way for the friction coefficient to be set using a time history function from the input file. If the `FRICION COEFFICIENT` line command is also present, the friction coefficient used by the code will be the `FRICION COEFFICIENT` value multiplied by the value obtained by the function.



8.14.5 Shell Overlap Removal Iterations

For contact with lofted geometry ACME has commands to remove overlap. Because the thickness of a shell must be preserved when shell lofting is requested, removing the initial overlap between nested shells becomes an iterative process whereby shell locations are adjusted to remove the overlap. This process is approximate and may not remove all the overlap in all cases. It is advised to check the corrected mesh to make sure that the mesh modifications are acceptable. For ACME two additional input lines may be needed in the `REMOVE INITIAL OVERLAP` command block to robustly remove overlap.

```

SHELL OVERLAP ITERATIONS = <integer>max_iter(10)
SHELL OVERLAP TOLERANCE = <real>shell_over_tol(0.0)

```

- **ACME Only:** The `SHELL OVERLAP ITERATIONS` command line controls the maximum number of iterations used by the overlap removal mechanism to resolve nested shells. By

default, the value of `max_iter` is 10. If the mesh has only a few layers of shells that may overlap, a value of 10 should suffice. However, if the mesh has a number of layers of shells that may overlap, this value may need to be larger.

- The `SHELL OVERLAP TOLERANCE` command line specifies an amount of overlap that is permitted to be left in the shell elements. This helps to limit the actual number of iterations required to remove the shell overlap, and to spread any remaining overlap over a number of shells instead of concentrating it all in a single shell. If the default value of 0.0 for the shell overlap tolerance is used, iteration continues until either all the overlap is removed or the maximum number of iterations is reached. If a nonzero value for the shell overlap tolerance is used, iteration continues until the tolerance is reached or the maximum number of iterations is reached. Note that the overlap removal process is only done once during an analysis, so a large number of iterations will only affect the first time step, not every time step.

The `SHELL OVERLAP ITERATIONS` and `SHELL OVERLAP TOLERANCE` commands have no meaning for analyses that do not have shell elements.

8.14.6 Surface Normal Smoothing

```
BEGIN SURFACE NORMAL SMOOTHING
  ANGLE = <real>angle_in_deg(60.0)
  DISTANCE = <real>distance(0.01)
  RESOLUTION = <string>NODE|EDGE(NODE)
END SURFACE NORMAL SMOOTHING
```

Surface normal smoothing is a feature that is primarily used in implicit contact.

Finite element discretization often results in models with faceted edges, while the true geometry of the part is actually smoothly curved. If the faces of adjacent finite elements on a surface have differing normals, the discontinuities at the edges between those faces can cause problems with contact. These discontinuities in the face normals are particularly troublesome with an implicit code such as Adagio, which uses an iterative solver to obtain a converged solution at every step. If a node is in contact near an edge with a normal discontinuity, the node may slide back and forth between the two neighboring faces during the iterations. Because the normal directions of the two faces differ, this can make it difficult to converge on a solution to this discontinuous contact problem.

Surface normal smoothing is a technique that creates a smooth variation in the normal near edges. The normal varies linearly from the value on one face to the value on the other face over a distance that spans the edge. A smoothly varying normal at the edge makes it easier for an iterative solver to obtain a converged solution in the case where a node has penetrated near the edge of a face.

Explicit dynamics does not use an iterative solver and thus does not encounter the difficulties associated with face normal discontinuities. Consequently, the `SURFACE NORMAL SMOOTHING` command block is not typically useful for explicit models, though is available in both explicit and implicit contact.

If the `SURFACE NORMAL SMOOTHING` command block is present, this feature is activated. There are three optional commands that can be used within this block to control the behavior of normal

smoothing.

- The `ANGLE` command is used to control whether smoothing should occur between neighboring faces. If the angle between two faces is less than the specified angle (given in degrees), smoothing is activated between them. Otherwise, the discontinuity is considered to be a feature of the model rather than an artifact of meshing, and they are not smoothed. The default value for `angle` is 60.
- The `DISTANCE` command specifies the distance as a fraction of the face size over which smoothing should occur. The specified value can vary from 0 to 1. The default value for `distance` is 0.01.
- The `RESOLUTION` command specifies the method used to determine the smoothed normal direction. The default `NODE` option uses a node-based algorithm to fit a smooth curve, while the `EDGE` option uses an edge-based algorithm.



8.14.7 Angle for Multiple Interactions

```
MULTIPLE INTERACTIONS = <string>ON|OFF (ON)  
MULTIPLE INTERACTIONS WITH ANGLE = <real>angle_in_deg(60.0)
```

When a node lies on the edge of a body, that node may need to support contact interactions with more than one face at the same time. For instance, see Figure 8.27. In Figure 8.27a, three blocks are shown, with a single node identified. Through contact, this node can interact with both the block on the upper right and the block on the bottom. If the node only supports a single interaction, then it will be arbitrarily considered for contact between one of the blocks, but not the other, as in Figure 8.27b. In this case, contact enforcement will prevent penetration into the lower block, but may permit penetration into the upper right block. The proper way to deal with this case is shown in Figure 8.27c, where multiple interactions are considered at the node.

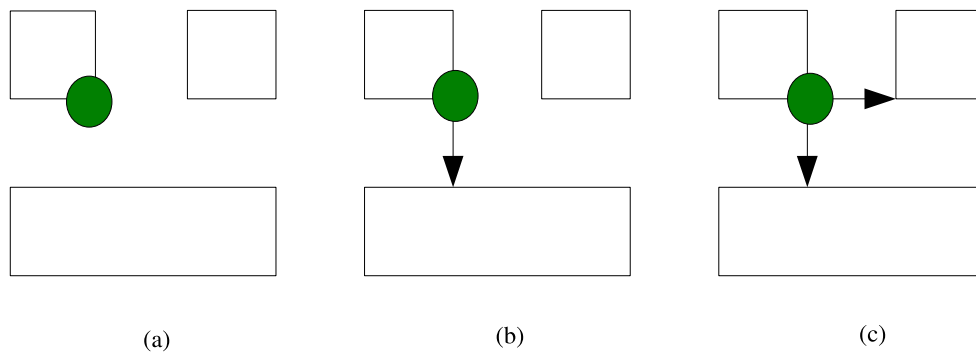


Figure 8.27: Illustrations of multiple interactions at a node: (a) initial configuration with node of interest identified (b) single interaction (c) multiple interactions

By default, Sierra/SM permits multiple interactions at a node. However, these multiple interactions may incur extra cost in the contact algorithm by increasing the number of interactions in

enforcement. Also, a local search algorithm (see Section 8.9.6), which uses various contact tracking approaches, may operate more efficiently when the node can only have one interaction. Finally, multiple interactions may lead to instabilities that can be eliminated by switching to single interactions. For these reasons, the `MULTIPLE INTERACTIONS` command line allows the user to choose whether multiple interactions should be considered at a node. A value of `OFF` indicates that a node can have only one interaction. This value affects all interactions in a contact definition. Sierra/SM does not currently have the capability to force single interactions for some surface pairs while allowing multiple interactions for other surface pairs.

When the `MULTIPLE INTERACTIONS` command line is `ON`, the number of interactions that can be considered at a node is dependent on the measure of curvature of those faces that are connected to the node. If the angle between two faces on which the node is attached is small, then only one interaction is allowed. However, in cases where the angle between the faces is large enough such that they form a discrete corner, multiple interactions are considered. The contact algorithms can properly handle only a limited number of interactions per node (currently three), so it is generally feasible to properly define interactions at a node, e.g., at the corner of a block.

The critical angle for multiple interactions is set with the `MULTIPLE INTERACTIONS WITH ANGLE` command line, where `angle` is the angle over which an edge is considered sharp. If the angle between adjoining faces is greater than this critical angle, multiple interactions can be created. By default, this critical angle is 60 degrees, which works well for most analyses. This value can be changed in the contact input if needed.

8.14.8 ACME Specific Lofted Geometry Options

ACME allows additional options in the `SEARCH OPTIONS` command block.

```
LOFTING ALGORITHM = <string>ON|OFF(ON)
COINCIDENT SHELL TREATMENT = <string>DISALLOW|IGNORE|
    SIMPLE(DISALLOW)
COINCIDENT SHELL HEX TREATMENT = <string>DISALLOW|
    IGNORE|TAPERED|EMBEDDED(DISALLOW)
```

In ACME contact the contact on shell elements can occur on either the meshed shell geometry, ignoring any shell thickness, or on the “lofted” geometry, including the thickness of the shell.

The `LOFTING ALGORITHM` command line determines whether contact on a shell should be done on the lofted geometry or on the original shell geometry. If the value of the `LOFTING ALGORITHM` command line is set to `ON`, shell contact uses the lofted geometry; if it set to `OFF`, shell contact uses the original shell geometry.

The `COINCIDENT SHELL TREATMENT` command line identifies how shells that share the same nodes should be treated. If the `DISALLOW` option is selected, (the default), then any time that shells in contact are detected to share all the same nodes, the code will abort with an error message indicating which elements were found to be coincident. The `DISALLOW` option should be used if you do not want any coincident shells to be considered in the analysis. The option operates essentially as a check on the mesh. If the `IGNORE` option is selected, any contact faces attached to coincident shells are ignored for contact. This option is only provided as a backup approach if

undiagnosed code problems arise from coincident shells. If such a case occurs, the `IGNORE` option may permit the user to continue with an analysis while the code team diagnoses the problem. The `SIMPLE` option enables coincident shells to be processed correctly. If lofting has been enabled and the `SIMPLE` option is selected, the thickness of the lofted coincident shell is taken as the largest thickness of all the coincident shells. If lofting is off and the `SIMPLE` option is selected, the coincident shell is treated as if only one of the shells is present.

The `COINCIDENT SHELL HEX TREATMENT` command line has a function similar to that of the `COINCIDENT SHELL TREATMENT` command line. The `COINCIDENT SHELL HEX TREATMENT` command line, however, identifies how shells that are fully coincident with the hexahedral elements are treated. If the `DISALLOW` option is selected (the default), then any time that a shell in contact is detected to share all the same nodes with the face of a continuum element, the code will abort with an error message indicating which elements were coincident. The `DISALLOW` option should be used if you do not want any shells coincident with hexahedra to be considered in the analysis. The option operates essentially as a check on the mesh. If the `IGNORE` option is selected, any contact faces attached to shells that are coincident with faces of continuum elements are ignored for contact. This option is only provided as a backup approach if undiagnosed code problems arise from coincident shells and continuum elements. If such a case occurs, the `IGNORE` option may permit the user to continue with an analysis while the code team diagnoses the problem. The `TAPERED` and `EMBEDDED` options permit shells that are coincident with faces of continuum elements to be processed in contact. The `TAPERED` option does two things: it includes for contact any faces that are on the free surface and ignores faces sandwiched between the shell and the continuum element, and it automatically adjusts the lofting of the surfaces to provide a smooth transition between shells that are not coincident with the faces of the continuum elements and those that are coincident with the faces of the continuum elements. The `EMBEDDED` option includes for contact both free surface faces and those that are between the coincident shells and faces of the continuum elements; the option does not adjust thicknesses to make smooth transitions between shells that are not coincident with faces of continuum elements and those that are coincident with faces of continuum elements. In general, the `TAPERED` option is preferred; only use the `EMBEDDED` option if the `TAPERED` option causes a code problem.

8.14.9 ACME Specific Debugging Options

When using ACME the following commands may be placed in the `BEGIN DEBUG` command block.

```
DEBUG NODE <int>nodeId
START AT STEP = <int>step
STOP AT STEP = <int>step
TIMERS = OFF|ON
```

`DEBUG NODE` is used to output everything that happens in contact related to node with ID `nodeId` starting at step `START AT STEP` and ending at step `STOP AT STEP`. The `TIMERS` command may be used to turn on ACME specific runtime timing information.

8.15 Analytic Rigid Surfaces



Warning: The commands for analytic rigid surface (ARS) contact in this section are in development and have not been tested for a wide variety of applications. See the Development Manual for a description of ARS contact.

```
BEGIN INTERACTION
  FRICTION DIRECTION = LEGACY|OPPOSITE
  NAME = <string>NAME
  RESET ACCUMULATED = OFF|ON
  SLIP PREDICTOR = <real>val
  SLIP TOLERANCE = <real>val
  SST FRICTION EVALUATION = LEGACY|MEAN QUADRATURE
  TYING TOLERANCE = <real>val
END
```

```
BEGIN SST FRICTION MODEL
  FRICTION SCALING = <real>
  MUVSSLIP FUNCTION = <string>
  PRESSURE EXPONENT = <real>
  PRESSURE REFERENCE = <real>
  SCALING FUNCTION = <string>
  XMAX = <real>
  XSTD = <real>
  YMAX = <real>
  YSTD = <real>
END
```

```
BEGIN TIME DEPENDENT FRICTION MODEL
  MU = <real>
  TIME FUNCTION = <string>
END
```

8.16 Deprecated Options



Warning: The commands in this section should not be used. They are related to legacy capabilities and are slated for eventual removal from the code base. This includes syntax that has been changed or commands that have been removed.

The following contact definition block options have been renamed

```
# Use ``SEARCH = `` instead of
CONTACT FORMULATION TYPE = ACME|DASH|ARS

# Use ``BEGIN SURFACE OPTIONS`` instead of
BEGIN SHELL LOFTING
....
END
```

The following are deprecated ACME options. The below command either have no effect or have been deemed to generally cause severe robustness issues in most analyses.

```
REINITIALIZE CONTACT ON RESTART = OFF|ON(OFF)
SMOOTH EDGE <real>distance SURFACE <surf>
UPDATE ALL SURFACES FOR ELEMENT DEATH = OFF|ON(ON)

BEGIN ENFORCEMENT OPTIONS
  MOMENTUM BALANCE TOLERANCE = <real>val
  ROTATION PRESERVING TIED ENFORCEMENT = OFF|ON
END

BEGIN INTERACTION
  ENFORCEMENT ALGORITHM = AUGMENTED_LAGRANGIAN|
    KINEMATIC|MOMENTUM_BALANCE
  # These options to interaction behavior are deprecated
  INTERACTION BEHAVIOR = <string>INFINITESIMAL_SLIDING|TIED
END

BEGIN SEARCH OPTIONS
  GLOBAL SEARCH INCREMENT = <int>num_steps
  PARALLEL CONSISTENCY = OFF|ON
  PARALLEL GHOSTING = NO_CULL|OVERLAP_CULL|SLAVE_CULL
  SECONDARY DECOMPOSITION = OFF|ON
END
```

The following commands in the BEGIN DASH OPTIONS block are no longer available. The explicit contact solution tolerance is now set automatically and needs no input from a command line.

The lofted sphere shape is now set in a `BEGIN SURFACE OPTIONS` command block. The lofted edges option was deemed not useful and this command has been removed from the code base.

```
BEGIN DASH OPTIONS
  GAP CONVERGENCE RATIO = <real>val
  LOFTED SPHERE REPRESENTATION = Cube|Icosahedron|Octahedron|Tetrahedron
  USE LOFTED SHELL EDGES = OFF|ON
END
```

References

- [1] K.H. Brown, R.M. Summers, M.W. Glass, A.S. Gullerud, M.W. Heinstein, and R.E. Jones. ACME: Algorithms for contact in a multiphysics environment, API version, 1.0. Technical Report SAND2001-3318, Sandia National Laboratories, Albuquerque, NM, 2001. [pdf](#).
- [2] M.W. Heinstein and T.E. Voth. Contact enforcement for explicit transient dynamics, 2005.
- [3] Sierra/SM Team. Sierra/SM Theory Manual. Technical Report SAND2013-4615, Sandia National Laboratory, Albuquerque, NM, 2013.

Chapter 9

Output

Sierra/SM produces a variety of output. This chapter discusses how to control the four major types of output:

- results output,
- history output,
- heartbeat output, and
- restart output.

The user is allowed to select a set of variables (internal, user-defined, or some combination thereof) for results output. If the user selects a nodal variable such as displacement for results output, the displacements for all the nodes in a model will be output to a results file. If the user selects an element variable such as stress for results output, the stress for all elements in the model that calculate this quantity (stress) will be output. The history output option lets the user select a specific set of information for output. For example, if the displacement at a particular node is critical, then the user may select only the displacement at that node as history output. The heartbeat output is similar to the history output except that the output is written to a text file instead of to a binary (exodusII [1]) file. The restart output is written so that any calculation can be halted at some arbitrary analysis time and then restarted at this time. The user has no control over what is written to the restart file. When a restart file is written, it must be a complete state description of the calculations at some given time. A restart file contains a great deal of information and is typically larger than a results file; thus the restart file output frequency must be chosen with care.

Section 9.1 describes the syntax for requesting output variables. Section 9.3 describes the results output. Included in the results output is a description of commands for user-defined output (Section 9.3.2). User-defined output lets the user post process analysis results as the code is running to produce a reduced set of output information. Section 9.4 describes the history output, Section 9.5 describes the heartbeat output, and Section 9.6 describes the restart output. All four types of output (results, history, heartbeat, and restart) can be synchronized for analyses with multiple regions. This scheduling functionality is discussed in Section 9.7. Section 9.8 describes the commands for performing output variable interpolation. Section 9.9 defines the output options for energy and rigid body global variables, which are defined at the region scope. Finally, in Section 9.10, there is a list of key variables. Unless otherwise noted, the command blocks and command lines discussed in Chapter 9 appear in the region scope.

9.1 Syntax for Requesting Variables

The basic method of requesting output is done by specifying the output type (nodal, element, or global), followed by the name of the variable. The original name will now appear in the output file, or the variable may be renamed using the `as` specifier.

```
var_type(element,nodal,global) <string>var_name
```

Variables may be accessed in the code either in whole or by component. Values at specific components or integration points of multi-component variables may be accessed via parenthesis syntax. Parenthesis syntax may be applied to results output, history output, element death, or any other command where variable names are specified. Values of single-component variables indexed in some other way may be accessed with underscore syntax, primarily applicable to rigid body fields as discussed in Section 9.1.4.

Parenthesis syntax is a variable name of the form:

```
<string>var_name(<index>component,<integer>integration_point)
```

For a variable named `var`, a variable name of the form `var(A,B)` asks for the `A` component of the variable at integration point `B`. If a variable is a vector, `x`, `y`, or `z` may be specified as the component. If a variable is 3x3 tensor, `xx`, `yy`, `zz`, `xy`, `xz`, `yz`, `yx`, `zx`, or `zy` may be specified as the component. Components for other types of variables may be requested through an integer index.

The characters `:` and `*` are wild cards if used for specifying either the component or the integration point. `var(:,B)` asks for all components of `var` at integration point `B`. `var(A,:)` asks for component `A` of `var` at all integration points. `var(:, :)` asks for all components of `var` at all integration points. `var` is shorthand for `var(:, :)`.

`var(A)` will behave slightly differently depending on the nature of the variable. If the variable has multiple components, then `var(A)` is treated like `var(A, :)`. If a variable has one and only one component then it is assumed that `A` refers to the integration point number rather than the component number and `var(A)` is treated like `var(1,A)`.

9.1.1 Example 1

Let stress be a tensor defined on a single integration point element and a displacement vector be defined at all model nodes. The following output variable specification:

```
element stress as str
nodal displacement as disp
```

asks for all the components of the stress tensor on elements and all components of the displacement vector on nodes. The code would write these variables to the output file:

```

str_xx
str_yy
str_zz
str_xy
str_xz
str_yz
disp_x
disp_y
disp_z

```

If only the *yy* component of stress is desired, either of the following could be used:

```

element stress(yy) as my_yy_str1
element stress(2) as my_yy_str2

```

If only the *z* component of displacement is desired either of the following could be specified:

```

nodal displacement(z) as my_z_disp1
nodal displacement(3) as my_z_disp2

```

Note, tensor index 2 corresponds to the *yy* component of the tensor and vector index 3 corresponds to the *z* component of the vector.

9.1.2 Example 2

Let *unrotated_stress* be a tensor defined at each of three integration points of an element (e.g., a shell element), and *eqps* a scalar material state variable also defined at each element integration point. To request all *unrotated_stress* tensor components and all *eqps* scalar data at all integration points the following could be specified:

```

element unrotated_stress as str
element eqps as eqps

```

Which would output the variables:

```

str_xx_1, str_yy_1, str_zz_1, str_xy_1, str_xz_1, str_yz_1
str_xx_2, str_yy_2, str_zz_2, str_xy_2, str_xz_2, str_yz_2
str_xx_3, str_yy_3, str_zz_3, str_xy_3, str_xz_3, str_yz_3
eqps_1, eqps_2, eqps_2

```

To request the *unrotated_stress* tensor and value of *eqps* at the second integration point, the following syntax can be used:

```

element unrotated_stress(:,2) as str_intg2
element eqps(2) as eqps_intg2

```

This would output the variables:

```
str_intg2_xx  
str_intg2_yy  
str_intg2_zz  
str_intg2_xy  
str_intg2_xz  
str_intg2_yz  
eqps_intg2
```

To request the `xy` component of `unrotated_stress` at all integration points any of the following could be used:

```
element  unrotated_stress(xy,*) as str_xy_all  
element  unrotated_stress(xy,:) as str_xy_all  
element  unrotated_stress(xy)   as str_xy_all
```

Any of the above would output:

```
str_xy_all_1  
str_xy_all_2  
str_xy_all_3
```

9.1.3 Other command blocks

The parenthesis syntax described above for results output can also be used in most other commands involving variable names. For example, to kill elements based on `yy` stress or `z` displacement the following could be specified:

```
begin element death  
  criterion is element value of stress(yy) > 1000  
  criterion is average nodal value of displacement(z) > 3.0  
end
```

9.1.4 Rigid Body Variables

Rigid body variables are provided as separate components of the rigid body fields, which may be accessed by outputting the desired field component(s). To access the field component of rigid bodies, an underscore syntax is employed (the parenthesis syntax applies to multi-component fields and integration points). The underscore syntax simply takes the desired field component and appends it with an underscore and the desired rigid body name.

For example the following lines in a history output block (see [Section 9.4](#))

```
global ax_rb1  
global displz_rb4
```


would output to the history file the variables

```
ax_rb1  
displz_rb4
```

which are the translational acceleration in the x -direction of rigid body `rb1` and the translational displacement in the z -direction of rigid body `rb4`, respectively (see Table [9.3](#)).

9.2 General Output Database Commands

A number of commands that control the format of the output database are available in several types of output (results, restart, history, and/or heartbeat). These commands are described in detail below.

```
DATABASE NAME = <string>results_file_name
DATABASE TYPE = <string>database_type(exodusII)
OVERWRITE = <string>OFF|ON|TRUE|FALSE|YES|NO
              (ON|TRUE|YES)
TITLE <string>user_title
PROPERTY = <string>propName = <string>propValue
```

The first two command lines (DATABASE NAME, DATABASE TYPE) give pertinent information about the database file. The command line

```
DATABASE NAME = <string>results_file_name
```

gives the name of the results file with the string `results_file_name`. If the file is to appear in the current directory and is named `job.e`, this command line would appear as:

```
DATABASE NAME = job.e}
```

If the file is to be created in some other directory, the command line must include the path to that directory. If the directory does not exist, all necessary directories will be created automatically.

Two special, shorthand meta-characters may be used in the results file name specification in the input deck: `%P` represents the number of processors in use for the analysis, while `%B` represents the base name of the analysis input file. If the `%P` character is used, for example, a specified database name of `results-%P/job.e` for a job running on N processors would be expanded to `results-N/job.e`, and the actual results files would be of the form `results-N/job.e.N.i`, where $i = \{0000, 0001, \dots, 1023\}$. If the `%B` character is used, for example, a results database name specified as `%B.e` in an input file named `my_analysis_run.i` and the `,` would yield a results output file named `my_analysis_run.e`. If no results database file name is given, the default file name is `%B.e`, the base name of the input file with the extension `.e`.

If the results file does not use the Exodus II format [1], the format for the results file must be specified using the command line:

```
DATABASE TYPE = <string>database_type(exodusII)
```

Both the Exodus II database and the XDMF database [2] are supported in Presto and Adagio. Exodus II is the preferred format, but other options may be added in the future.

The `OVERWRITE` command line can be used to prevent the overwriting of existing results files.

```
OVERWRITE = <string>OFF|ON|TRUE|FALSE|YES|NO (ON|TRUE|YES)
```

The `OVERWRITE` command line allows only a single value. The options `FALSE`, `NO`, or `OFF` will cause the code to terminate before existing results files can be overwritten, while `TRUE`, `YES`, or

ON will cause existing results files to be overwritten by new files (the default status). Suppose, for example, that an existing results is named file named `job21.e`, and suppose that an input file with a `RESULTS OUTPUT` command block containing

```
OVERWRITE = ON
DATABASE NAME = job21.e
```

is run. In this case the existing file `job21.e` will be overwritten.

Whether or not files are overwritten is also impacted by the use of the automatic read and write option for restart files described in Section 9.6.1.2. If using the automatic read and write option for restart files, the results files, like the restart files, are automatically managed. The automatic read and write option in restart adds extensions to file names and prevents the overwriting of any existing restart or results files. For the case of a user-controlled read and write of restart files (Section 9.6.1.3) or of no restart, however, the `OVERWRITE` command line is useful for preventing the overwriting of results files.

A title may be added to the results file with the `TITLE` command line. The specified `user_title` will be written to the ‘title’ field of the results file. Some of the programs that process the results file (such as various SEACAS programs [3]) can read and display this information.

The `PROPERTY` command can change some of the fundamental properties of the output database. For example, maximum variable name length, which, in Exodus II format, has a limit of 100 characters. Note, changing this properties is not recommended unless by direction of the development team.

9.3 Results Output

The results output capability allows the user to select some set of variables to be written to a file at various time intervals. All values for each selected variable will be written to the results file. The interval at which information is written can be changed throughout the analysis time. The name of the results file is set in the `RESULTS OUTPUT` command block.

9.3.1 Exodus Results Output File

```
BEGIN RESULTS OUTPUT <string>results_name
#
# Database Commands
DATABASE NAME = <string>results_file_name
DATABASE TYPE = <string>database_type(exodusII)
OVERWRITE = <string>OFF|ON|TRUE|FALSE|YES|NO
            (ON|TRUE|YES)
TITLE <string>user_title
PROPERTY <string>propName = <string>propVal
TIMESERIES NAME = <string>timeseries_file_name

NODE VARIABLES = <string>variable_name
                [AS <string>dbase_variable_name] ...
                <string>variable_name [AS
                <string>dbase_variable_name]
                | NODAL VARIABLES = <string>variable_name
                [AS <string>dbase_variable_name] ...
                <string>variable_name [AS
                <string>dbase_variable_name]

NODESET VARIABLES = <string>variable_name
                  [AS <string>dbase_variable_name] ...
                  <string>variable_name [AS
                  <string>dbase_variable_name]
                  | NODESET VARIABLES = <string>variable_name
                  [AS <string>dbase_variable_name]
                  INCLUDE|ON|EXCLUDE <string list>nodelist_names
                  ... <string>variable_name
                  [AS <string>dbase_variable_name] INCLUDE|ON|EXCLUDE
                  <string list>nodelist_names

SURFACE VARIABLES = <string>variable_name
                  [AS <string>dbase_variable_name] ...
                  <string>variable_name [AS
                  <string>dbase_variable_name]
                  | SURFACE VARIABLES = <string>variable_name
                  [AS <string>dbase_variable_name]
                  INCLUDE|ON|EXCLUDE <string list>surface_names
                  ... <string>variable_name
```

```

        [AS <string>dbase_variable_name] INCLUDE|ON|EXCLUDE
        <string list>surface_names

FACE VARIABLES = <string>variable_name
    [AS <string>dbase_variable_name] ...
    <string>variable_name [AS
    <string>dbase_variable_name]
    | FACE VARIABLES = <string>variable_name
        [AS <string>dbase_variable_name]
        INCLUDE|ON|EXCLUDE <string list>surface_names
        ... <string>variable_name
        [AS <string>dbase_variable_name] INCLUDE|ON|EXCLUDE
        <string list>surface_names

ELEMENT VARIABLES = <string>variable_name
    [AS <string>dbase_variable_name] ...
    <string>variable_name [AS
    <string>dbase_variable_name]
    | ELEMENT VARIABLES = <string>variable_name
        [AS <string>dbase_variable_name]
        INCLUDE|ON|EXCLUDE <string list>block_names
        ... <string>variable_name
        [AS dbase_variable_name] INCLUDE|ON|EXCLUDE
        <string list>block_names

GLOBAL VARIABLES = <string>variable_name
    [AS <string>dbase_variable_name] ...
    <string>variable_name [AS
    <string>dbase_variable_name]
OUTPUT MESH = REFINED|EXPOSED SURFACE (REFINED)
COMPONENT SEPARATOR CHARACTER = <string>character|NONE
INCLUDE = <string>list_of_included_entities
EXCLUDE = <string>list_of_excluded_entities
START TIME = <real>output_start_time
TIMESTEP ADJUSTMENT INTERVAL = <integer>steps
AT TIME <real>time_in INCREMENT =
    <real>time_increment_dt
AT STEP <integer>step_begin INCREMENT =
    <integer>step_increment
ADDITIONAL TIMES = <real>output_time1
    <real>output_time2 ...
ADDITIONAL STEPS = <integer>output_step1
    <integer>output_step2 ...
TERMINATION TIME = <real>termination_time_value
SYNCHRONIZE_OUTPUT
USE OUTPUT SCHEDULER <string>scheduler name
OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|

```

➡ **Explicit**

```

SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|
SIGKILL|SIGILL|SIGSEGV
END [RESULTS OUTPUT <string>results_name]

```

See Section 9.2 for details on the general database commands such as DATABASE NAME, DATABASE TYPE, OVERWRITE, etc. The command TIMESERIES NAME optionally emits a time-series file containing the list of root output database filenames as they are written. This is useful for simulations on high numbers of processors with many mesh modifications that output a large number of files.

A results file, the results to be included in this file, and the frequency at which results are written may be specified using a RESULTS OUTPUT command block. The command block appears inside the region scope. More than one results file can be specified for an analysis; thus, for each results file, there will be one RESULTS OUTPUT command block. The command block begins with:

```
BEGIN RESULTS OUTPUT <string>results_name
```

and is terminated with:

```
END [RESULTS OUTPUT <string>results_name]
```

where results_name is a user-selected name for the command block. Nested within the RESULTS OUTPUT command block is a set of command lines, as shown in the block summary given above.

The other command lines that appear in the RESULTS OUTPUT command block determine the type and frequency of output information. Descriptions of these command lines follow in Section 9.3.1.1 through Section 9.3.1.12.

9.3.1.1 Output Nodal Variables

```

NODE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]
| NODAL VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]

```

Any nodal variable in Sierra/SM can be selected for output in the results file by using a command line in one of the two forms shown above. The only difference between the two forms is the use of NODE VARIABLES or NODAL VARIABLES. The string variable_name is the name of the nodal variable to output. The string variable_name can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4). Any nodal variable requested for output is output for all nodes.

An alias can be specified for any of the nodal variables by using the AS specification. For example, to output the external forces in Sierra/SM, which are defined as force_external, with the alias f_ext, the command line

```
NODE VARIABLES = force_external AS f_ext
```

would be used. In this example, the external force is a vector quantity. For a vector quantity at a node, suffixes are appended to the variable name (or alias name) to denote each vector component. The results database would have three variable names associated with the external force: `f_ext_x`, `f_ext_y`, and `f_ext_z`. The component separator—an underscore in this example—may be changed using the `COMPONENT SEPARATOR CHARACTER` command line (see Section 9.3.1.7).

The `NODE VARIABLES` command line can be used an arbitrary number of times within a `RESULTS OUTPUT` command block. It is also possible to specify more than one nodal variable for output on a command line, as indicated by the ellipsis in the command line format. In the next example, two nodal variables are specified for output. Internal forces are defined as `force_internal`.

```
NODE VARIABLES = force_external force_internal
```

Aliases can be optionally specified for each of the variables in a single command line. For example, to output the alias `f_ext` for external forces and to output the alias `f_int` for internal forces, the command line

```
NODE VARIABLES = force_external AS f_ext
                  force_internal AS f_int
```

9.3.1.2 Output Node Set or Surface Variables

```
NODESET VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]
| NODESET VARIABLES = <string>variable_name
[AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>nodelist_names
... <string>variable_name
[AS <string>dbase_variable_name] INCLUDE|ON|EXCLUDE
<string list>nodelist_names
SURFACE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]
| SURFACE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>surface_names
... <string>variable_name
[AS <string>dbase_variable_name] INCLUDE|ON|EXCLUDE
<string list>surface_names
```

A nodal variable may be defined on a subset of the total set of nodes defining a model or a face variable may be defined on a subset of surfaces. A nodal variable defined only on a subset of nodes is called a node set variable. A face variable defined only on a subset of surfaces is called a surface variable.

The first form of this command line is as follows:

```
NODESET|SURFACE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS <string>dbase_variable_name]
```

Here, the string `variable_name` is a variable associated with one or more node sets or surfaces. In this form, the variable is output for all node sets or surfaces associated with that variable.

An alias can be specified in the results file for any of the variables by using the `AS` option. Suppose, for example, the node set variable `force_nsetype` is to be output with the name `fnsetype` in the results file. The corresponding command line is:

```
NODESET VARIABLES = force_nsetype AS fnsetype
```

The `NODESET|SURFACE VARIABLES` command line can be used an arbitrary number of times within a `RESULTS OUTPUT` command block. It is also possible to specify more than one node set variable for output on a command line, as indicated by the ellipsis in the command line format. In the next example, two node set variables are specified for output. Here, the second node set variable is defined as `force_nsetype2`.

```
NODESET VARIABLES = force_nsetype force_nsetype2
```

Aliases can be optionally specified for each of the variables in a single command line. For example, to output the alias `fnsetype` for node set variable `force_nsetype` and also to output the alias `fnsetype2` for node set variable `force_nsetype2`, the command line

```
NODESET VARIABLES = force_nsetype AS fnsetype
                    force_nsetype2 AS fnsetype2
```

would be used.

The second form of the command line is as follows:

```
NODESET VARIABLES = <string>variable_name
[AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>nodelist_names
... <string>variable_name [AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>nodelist_names
SURFACE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>surface_names
... <string>variable_name [AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>surface_names
```

This form of the `NODESET|SURFACE VARIABLES` command line is similar to the first, except that the user can control which node sets are used for output. The user can include a specific list of node sets or surfaces for output by using the `INCLUDE` option or the `ON` option. (The keyword

INCLUDE is synonymous with the keyword ON.) Alternatively, the user can exclude a specific list of node sets for output by using the EXCLUDE option.

Suppose that the node set variable `force_nsetype` from the above example has been defined for `nodelist_10`, `nodelist_11`, `nodelist_20`, and `nodelist_21`. If we only want to output the node set variable for node sets `nodelist_10`, `nodelist_11`, and `nodelist_21`, then we could specify the NODESET command line as follows:

```
NODESET VARIABLES = force_nsetype AS fnsetype
INCLUDE nodelist_10, nodelist_11, nodelist_21
```

(In the above command line, the keyword ON could be substituted for INCLUDE.) Alternatively, we could use the command line:

```
NODESET VARIABLES = force_nsetype AS fnsetype
EXCLUDE nodelist_20
```

In the above command lines, an alias for a node set can be substituted for a node set identifier. For example, if `center_case` is an alias for `nodelist_10`, then the string `center_case` could be substituted for `nodelist_10` in the above command lines. Because a node set identifier is a mesh entity, the alias for the node set identifier would be defined via an ALIAS command line in a FINITE ELEMENT MODEL command block.

The list of identifiers uses a comma to separate one node set identifier from the next node set identifier.

9.3.1.3 Output Face Variables

```
FACE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]
| FACE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>surface_names
... <string>variable_name
[AS <string>dbase_variable_name] INCLUDE|ON|EXCLUDE
<string list>surface_names
```

A variable may be defined on some set of faces that constitute a surface. A variable defined on a set of faces is called a face variable. The FACE VARIABLES command line allows the user to specify a face variable for output to the results file. Two forms of the FACE VARIABLES command line exist and either form will output a face variable. The first form of the command line is as follows:

```
FACE VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]
```

Here, the string `variable_name` is a face variable associated with one or more surfaces. In this form, the face variable is output for all surfaces associated with that face variable.

An alias can be specified in the results file for any face variable by using the `AS` option. For example, to output a face variable `pressure_face` with the name `pressuref` in the results file, the command line

```
FACE VARIABLES = pressure_face AS pressuref
```

would be used. The `FACE VARIABLES` command line can be used an arbitrary number of times within a `RESULTS OUTPUT` command block. It is also possible to specify more than one face variable for output on a command line, as indicated by the ellipsis in the command line format. In the next example, two face variables are specified for output. Here, the second face variable is defined as `scalar_face2`:

```
FACE VARIABLES = pressure_face scalar_face2
```

Aliases can be optionally specified for each of the variables in a single command line. For example, to output the alias `pressuref` for face variable `pressure_face` and also to output the alias `scalarf2` for face variable `scalar_face2`, the command line

```
FACE VARIABLES = pressure_face AS pressuref
                  scalar_face2 AS scalarf2
```

The second form of the command line is as follows:

```
FACE VARIABLES = <string>variable_name
                  [AS <string>dbase_variable_name]
                  INCLUDE|ON|EXCLUDE <string list>surface_names
                  ... <string>variable_name
                  [AS <string>dbase_variable_name] INCLUDE|ON|EXCLUDE
                  <string list>surface_names
```

This form of the `FACE VARIABLES` command line is similar to the first, except that the user can control which surfaces are used for output. The user can include a specific list of surfaces for output by using the `INCLUDE` option or the `ON` option. (The keyword `INCLUDE` is synonymous with the keyword `ON`.) Alternatively, the user can exclude a specific list of surfaces for output by using the `EXCLUDE` option.

Suppose that the face variable `pressure_face` from the above example has been defined for `surface_10`, `surface_11`, `surface_20`, and `surface_21`. If we only want to output the face variable for `surface_10`, `surface_11`, and `surface_21`, then we could specify the `FACE VARIABLES` command line as follows:

```
FACE VARIABLES = pressure_face AS pressuref
                  INCLUDE surface_10, surface_11,
                  surface_21
```

(In the above command line, the keyword `ON` could be substituted for `INCLUDE`.) Alternatively, we could use the command line:

```
FACE VARIABLES = pressure_face AS pressuref
EXCLUDE surface_20
```

In the above command lines, an alias for a surface can be substituted for a surface identifier. For example, if `center_case` is an alias for `surface_10`, then the string `center_case` could be substituted for `surface_10` in the above command lines. Because a surface identifier is a mesh entity, the alias for the surface identifier would be defined via an `ALIAS` command line in a `FINITE ELEMENT MODEL` command block.

The list of identifiers uses a comma to separate one surface identifier from the next surface identifier.

9.3.1.4 Output Element Variables

```
ELEMENT VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]
| ELEMENT VARIABLES = <string>variable_name
[AS <string>dbase_variable_name]
INCLUDE|ON|EXCLUDE <string list>block_names
... <string>variable_name
[AS dbase_variable_name] INCLUDE|ON|EXCLUDE
<string list>block_names
```

Any element variable in Sierra/SM can be selected for output in the results file by using the `ELEMENT VARIABLES` command line. Two forms of the `ELEMENT VARIABLES` command line exist and either form will output an element variable. The first form of these command lines are as follows:

```
ELEMENT VARIABLES = <string>variable_name
[AS <string>dbase_variable_name] ...
<string>variable_name [AS
<string>dbase_variable_name]
```

Here, the string `variable_name` is the name of the element variable to output. The string `variable_name` can be a variable listed in [Section 9.10](#) or a user-defined variable (see [Section 9.3.2](#) and [Section 11.2.4](#)).

In the first form of the `ELEMENT VARIABLES` command line, the element variable is output for all element blocks that have the element variable as a defined variable. For example, all the solid elements have `stress` as a defined variable. If output of the element variable `stress` were requested on a mesh consisting of hexahedral and tetrahedral elements, then `stress` would be output for all element blocks consisting of hexahedral and tetrahedral elements.

An alias can be specified for any of the element variables by using the `AS` specification. For example, to output the stress in Sierra/SM, which is defined as `stress`, with the alias `str`, the command line

```
ELEMENT VARIABLES = stress AS str
```

would be used. In this example, stress is a symmetric tensor quantity. For a symmetric tensor quantity, suffixes are appended to the variable name (or alias name) to denote each symmetric tensor component. The results database would have six variable names associated with the stress: stress_xx, stress_yy, stress_zz, stress_xy, stress_yz, and stress_zx. The tensor component separator, an underscore in this example, may be changed by using the COMPONENT SEPARATOR CHARACTER command line (see Section 9.3.1.7).

The ELEMENT VARIABLES command line can be used an arbitrary number of times within a RESULTS OUTPUT command block. It is also possible to specify more than one element variable for output on a command line, as indicated by the ellipsis in the command line format. In the following example, two element variables are specified for output. Here, the second element variable is defined as left_stretch.

```
ELEMENT VARIABLES = stress left_stretch
```

Aliases can be optionally specified for each of the variables in a single command line. For example, to output the alias str for element variable stress and also to output the alias strch for face variable left_stretch, the command line

```
ELEMENT VARIABLES = stress AS str
                    left_stretch AS strch
```

The second form of the command line is as follows:

```
ELEMENT VARIABLES = <string>variable_name
                    [AS <string>dbase_variable_name]
                    INCLUDE|ON|EXCLUDE <string list>block_names
                    ... <string>variable_name
                    [AS dbase_variable_name] INCLUDE|ON|EXCLUDE
                    <string list>block_names
```

This form of the ELEMENT VARIABLES command line is similar to the first, except that the user can control which element blocks are used for output. The user can include a specific list of element blocks for output by using the INCLUDE option or the ON option. (The keyword INCLUDE is synonymous with the keyword ON.) Alternatively, the user can exclude a specific list of element blocks for output by using the EXCLUDE option.

Suppose the element variable stress from the above example exists for element blocks block_10, block_11, block_20, and block_21. If we only want to output the element variable for block_10, block_11, and block_21, then we could specify the ELEMENT VARIABLES command line as follows:

```
ELEMENT VARIABLES = stress AS str
                    INCLUDE block_10, block_11,
                    block_21
```

(In the above command line, the keyword ON could be substituted for INCLUDE.) Alternatively, we could use the command line:

```
ELEMENT VARIABLES = stress AS str
    EXCLUDE block_20
```

In the above command lines, an alias for an element block can be substituted for an element block identifier. For example, if `center_case` is an alias for `block_10`, then the string `center_case` could be substituted for `block_10` in the above command lines. Because an element block identifier is a mesh entity, the alias for the element block identifier would be defined via an `ALIAS` command line in a `FINITE ELEMENT MODEL` command block. The list of identifiers uses a comma to separate one element block identifier from the next element block identifier.

For multi-integration point elements, quantities from the integration points are appended with a numerical index indicating the integration point. A suffix ranging from 1 to the number of integration points is attached to the quantity to indicate the corresponding integration point. The suffix is padded with leading zeros. If the number of integration points is less than 10, the suffix has the form `_i`, where *i* ranges from 1 to the number of integration points. If the number of integration points is greater than or equal to 10 and less than 100, the sequence of suffixes takes the form `_01`, `_02`, `_03`, and so forth. Finally, if the number of integration points is greater than or equal to 100, the sequence of suffixes takes the form `_001`, `_002`, `_003`, and so forth. As an example, if the von Mises stress is requested for a shell element with 15 integration points, then the quantities `von_mises_01`, `von_mises_02`, ..., `von_mises_15` are output for the shell element.

In the above discussion concerning the format for output at multiple integration points, the underscore character preceding the integration point number can be replaced by another delimiter or the underscore character can be eliminated by use of the `COMPONENT SEPARATOR CHARACTER` command line (see Section 9.3.1.7).

Shell tensor quantities `transform_shell_stress`, `transform_shell_strain` and `transform_shell_rate_of_deformation` may be transformed to a user specified shell local co-rotational coordinate system (an in-plane coordinate system that rotates with the shell element) for output using the `ORIENTATION` shell section command. If no orientation is specified, these in-plane stresses and strains are output in the default orientation. See Section 6.2.4 for more details.

9.3.1.5 Subsetting of Output Mesh

A specified subset of the entities (element blocks, nodesets, surfaces) in the mesh can be output to the results database using the `INCLUDE` or `EXCLUDE` commands. The syntax is:

```
INCLUDE = <string>list_of_included_entities
EXCLUDE = <string>list_of_excluded_entities
```

Either command can appear multiple times within the results output block, but a specific entity cannot be specified in both. If the `INCLUDE` command is specified, the results database will only contain the listed entities of that type; if the `EXCLUDE` command is specified, the results database will contain all entities of the type listed except for the listed entities. If the model has surfaces or nodesets, only the portion of the surfaces or nodesets on the selected element blocks will be output.

9.3.1.6 Output Mesh Selection

```
OUTPUT MESH = REFINED|EXPOSED SURFACE (REFINED)
```

The `OUTPUT MESH` command provides a way to reduce the amount of data written to the results database. There are two options that can be selected:

REFINED Outputs the full 3D model in the current mesh configuration.

EXPOSED SURFACE Only output the element faces that make up the ‘skin’ of the finite element model; no internal nodes or elements will be written to the results database. The element results variables will be applied to the skin faces. If the mesh is visualized without any cutting planes, the display should look the same as if the original full mesh were visualized; however, the amount of data written to the output file can be much less than is needed if the full mesh were output.

9.3.1.7 Component Separator Character

```
COMPONENT SEPARATOR CHARACTER = <string>character|NONE
```

The component separator character is used to separate an output-variable base name from any suffixes. For example, the variable `stress` can have the suffixes `xx`, `yy`, etc. By default, the base name is separated from the suffixes with an underscore character so that we have `stress_xx`, `stress_yy`, etc. in the results output file.

The underscore can be replaced as the default separator by using the above command line. For example, to use a period (.) as the separator, the command line

```
COMPONENT SEPARATOR CHARACTER = .
```

would be used. For the `stress` output example, stress components would then appear in the results output file as `stress.xx`, `stress.yy`, etc. For shell element blocks, an integration point suffix preceded, by default, with an underscore is also present; the above command line also resets this separator character to period (.). In the `stress` output example, in this case the results file would contain `stress.xx.01`, `stress.xx.02`, etc., for the shell elements. The separator can be eliminated with an empty string or `NONE`.

9.3.1.8 Output Global Variables

```
GLOBAL VARIABLES = <string>variable_name  
                  [AS <string>dbase_variable_name] ...  
                  <string>variable_name [AS  
                  <string>dbase_variable_name]
```

Any global variable in Sierra/SM can be selected for output in the results file by using the `GLOBAL VARIABLES` command line. The string `variable_name` is the name of the global variable. The string `variable_name` can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4).

Kinetic, external, hourglass, internal energies and momentum¹ can be requested as the sum over the entire mesh or as sums over element blocks as noted in Section 9.10. For example, if the mesh contains element blocks with IDs 100 and 200, the kinetic and hourglass energy summed over each of these blocks can be requested with the commands

```
GLOBAL VARIABLES = ke_block100 as ke100
GLOBAL VARIABLES = ke_block200 as ke200
GLOBAL VARIABLES = hge_block100 as hge100
GLOBAL VARIABLES = hge_block200 as hge200
# This outputs the components of the Momentum per block
GLOBAL VARIABLES = momentum_block100_x as mx100
GLOBAL VARIABLES = momentum_block100_y as my100
GLOBAL VARIABLES = momentum_block100_z as mz100
```

and the total kinetic energy as

```
GLOBAL VARIABLES = kinetic_energy as ke
```

Kinetic, internal, and external energies are computed as nodal values. Since nodes may be shared by element blocks, the total sum of an energy quantity over individual element blocks will not necessarily be equal to the global sum requested using the `kinetic_energy`, `internal_energy`, or `external_energy` variable names.

With the `AS` keyword, the variable and an alias for this variable may be optionally specified in the results file. For example, to output the time steps in Sierra/SM (`timestep`) with the alias `tstep`, the command line

```
GLOBAL VARIABLES = timestep AS tstep
```

would be used. The `GLOBAL VARIABLES` command line can be used an arbitrary number of times within a `RESULTS OUTPUT` command block. It is also possible to specify more than one global variable for output on a command line. If output of the CPU time (`cpu_time`) is also desired for output with the alias `cpu`, the command line

```
GLOBAL VARIABLES = timestep as tstep cpu_time as cpu
```

should be used.

9.3.1.9 Specifying Time Steps for Output

```
START TIME = <real>output_start_time
TERMINATION TIME = <real>output_termination_time
AT TIME <real>time_begin INCREMENT = <real>time_increment_dt
AT STEP <integer>step_begin INCREMENT = <integer>step_increment
```

¹See Chapter 8 of the Sierra/SM Theory Manual [4] for more on momentum and energy balance. Internal energy includes both elastic and dissipated energy and contact energy is included in the external energy. Significant error in the energy balance (see Section 3.1.5) often indicates the problem is under-resolved.

```

ADDITIONAL TIMES = <real>output_time1 <real>output_time2 ...
ADDITIONAL STEPS = <integer>output_step1 <integer>output_step2 ...
TIMESTEP ADJUSTMENT INTERVAL = <integer>steps

```

Explicit

Several options are available to define output; output steps will be included in a results, history, or heartbeat output file. Note, these commands control only what time steps will be output, not what time steps will be computed. Thus requesting output at some time N will trigger an output of the next computed time step that is equal to or greater than time N . The analysis time stepping setup controls what time steps are computed, see Sections 3.1 and 4.10.

Using the `START TIME` command line, results are output to file beginning at time `output_start_time`. This command is optional, but if present, no results will be written before the `output_start_time`. If other commands set times for results output (e.g., `AT TIME`, `ADDITIONAL TIMES`) that are less than `output_start_time`, those times will be ignored, and results will not be written at those times.

If the `TERMINATION TIME` command is used, results will not be written to the results file after time `output_termination_time`. If other commands set times for results such (`AT TIME`, `ADDITIONAL TIMES`) that are greater than `termination_time_value`, those times will be ignored, and results will not be written at those times.

The `AT TIME` command will trigger output starting at the time specified by `time_begin` as long as this time is greater than `output_start_time` specified in a `START TIME` command, and then at every subsequent time increment `time_increment_dt` thereafter. Output will also occur at the end of the analysis even if it does not fall exactly at one of the specified intervals.

The `AT STEP` command will trigger output starting at the step specified by `step_begin` and then at every subsequent step increment `step_increment` thereafter. Output will also occur at the end of the analysis even if it does not fall exactly at a specified interval.

If both the `AT TIME` and the `AT STEP` commands are used time steps that match either condition will be output. For example the following would output time steps 0.0001, 0.0002, 0.0003, etc, as well as every hundredth step.

```

at time 0.0, increment = 0.0001
at step 0, increment = 100

```

If multiple `AT TIME` or `AT STEP` commands are used the commands define different output frequencies at different time intervals. For example:

```

at time 0.00, increment = 0.25
at time 1.00, increment = 0.01
at time 1.05, increment = 0.25

```

From time 0.00 to 1.00 the steps will output at time interval 0.25, at time 1.00 the output interval will switch to 0.01 and then the output interval will switch back to 0.25 at time 1.05. Specifically this will output the following time steps 0.00, 0.25, 0.50, 0.75, 1.00,

1.01, 1.02, 1.03, 1.04, 1.05, 1.30, 1.55, etc.. Use of multiple AT TIME or AT STEP commands is often used to obtain high frequency of output at some critical analysis time while still limiting the overall size of the results file.

The ADDITIONAL TIMES command line can be used to specify a number of additional output times. If output_start_time is specified in a START TIME command, then no additional times that are less than output_start_time will output. Output will also occur at the end of the analysis even if it does not fall exactly at a specified interval.

The ADDITIONAL STEPS command line can be used to specify output at a number of other analysis steps. If output_start_time is specified in a START TIME command, then no additional steps at times less than output_start_time will output. Output will also occur at the end of the analysis even if it does not fall exactly at a specified interval.

The TIMESTEP ADJUSTMENT INTERVAL command line is used to specify that the output will be at exactly the times specified. To hit the output times exactly in an explicit, transient dynamics code, it is necessary to adjust the time step as the time approaches an output time. The integer value steps in the TIMESTEP ADJUSTMENT INTERVAL command line specifies the number of time steps to look ahead to adjust the time step. This command is only functional in explicit analysis. If this command line does not appear, results are output at times closest to the specified output times.



Examples The following examples demonstrate how time step output works for many cases. For this example assume the analysis time stepping is setup to compute the following steps:

step	1	2	3	4	5	6	7	8	9	10	11
time	0.0	0.5	1.0	1.5	2.0	2.5	3.0	3.5	4.0	4.5	5.0

1. at step 0, increment = 1

This will output all 11 computed time steps.

2. at step 1, increment = 4
additional steps = 6 7

This will output every third time step, steps 6 and 7, and the final time step, i.e., steps 1, 5, 6, 7, 9, and 11.

3. additional times 1.6

This will output at the next computed time t after the specified time $t_o = 1.6$, so output will be written only at step 5, at time $t = 2.0 > t_o$.

4. at time 0.0, increment = 1.3

This will output every 1.3 seconds of analysis times, i.e., times 0.0, 1.3, 2.6, 3.9, and 5.2. For a specified output time t_o not coincident with actual simulation time steps $\{t_s\}$, the next simulation time step $t \in \{t_s\}$, $t \geq t_o$ will be output. The actual output steps will then be as follows:

step	1	4	7	9
time	0.0	1.5	3.0	4.0

5. at step 0, increment = 1
start time = 2.0
end time = 4.0

This will output at every step between the specified start and end times, i.e., at steps 5, 6, 7, 8, and 9.

9.3.1.10 Synchronize Output

SYNCHRONIZE OUTPUT

In an analysis with multiple regions, it is sometimes desirable to synchronize the output of results data between the regions. This can be done by adding the `SYNCHRONIZE OUTPUT` command line to the results output block. If a results block has this set, then it will write output whenever a previous region writes output. The ordering of regions is based on the order in the input file, algorithmic considerations, or by solution control specifications.

Although the `USE OUTPUT SCHEDULER` command line can also synchronize output between regions, the `SYNCHRONIZE OUTPUT` command line will synchronize the output with regions where the output frequency is not under the direct control of the Sierra IO system. Examples of this are typically coupled applications where one or more of the codes are not Sierra-based applications such as Alegria and CTH. A results block with `SYNCHRONIZE OUTPUT` specified will also synchronize its output with the output of the external code.

The `SYNCHRONIZE OUTPUT` command can be used with other output scheduling commands such as time-based or step-based output specifications.

9.3.1.11 Use Output Scheduler

USE OUTPUT SCHEDULER <string>scheduler_name

In an analysis with multiple regions, it can be difficult to synchronize output such as results files. To help synchronize output for analyses with multiple regions, an `OUTPUT SCHEDULER` command block may be specified at the `SIERRA` scope. The scheduler can then be referenced in the `RESULTS OUTPUT` command block via the `USE OUTPUT SCHEDULER` command line. The string `scheduler_name` must match a name used in an `OUTPUT SCHEDULER` command block. See Section 9.7 for a description of using this command block and the `USE OUTPUT SCHEDULER` command line.

9.3.1.12 Write Results If System Error Encountered

OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|
SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|
SIGKILL|SIGILL|SIGSEGV

The `OUTPUT ON SIGNAL` command line is used to initiate the writing of a results file when the system encounters a system error. Only one error type in the list of error types should be entered for this command line. These system errors cause the code to terminate before the code can add any current results output (results output past the last results output time step) to the results output file. If the code encounters the specified error type during execution, a results file will be written before execution is terminated. This command line can also be used to force the writing of a results file at some point during execution of the code. Suppose the command line

```
OUTPUT ON SIGNAL = SIGUSR2
```

is included in the input file. While the code is running, a user can execute (from the keyboard) the system command line

```
kill -s SIGUSR2 pid
```

to terminate execution and force the writing of a results file. In the above system command line, *pid* is the process identifier, an integer. The `OUTPUT ON SIGNAL` command line is primarily a debugging tool for code developers.

9.3.2 User-Defined Output

```
BEGIN FILTER <string>filter_name
  ACOEFF = <real_list>a_coeff
  BCOEFF = <real_list>b_coeff
  INTERPOLATION TIME STEP = <real>ts
END [FILTER]

BEGIN USER OUTPUT
  # mesh-entity set commands
  NODE SET = <string list>nodelist_names
  SURFACE = <string list>surface_names
  BLOCK = <string list>block_names
  INCLUDE ALL BLOCKS
  REMOVE NODE SET = <string list>nodelist_names
  REMOVE SURFACE = <string list> surface_names
  REMOVE BLOCK = <string list>block_names
  #
  # compute result commands
  PROCESS ELEMENTS = ACTIVE_ONLY|INACTIVE_ONLY|ALL (ACTIVE_ONLY)
  COMPUTE GLOBAL <string>result_var_name
    AS SUM|MAX|MIN|MAX ABSOLUTE VALUE|L2NORM|VARIANCE
    OF NODAL|ELEMENT|FACE <string>source_var_name
  COMPUTE GLOBAL <string>result_var_name
    AS AVERAGE
    OF NODAL|ELEMENT|FACE <string>source_var_name
    [WEIGHTED BY <string>weight_var_name]
  COMPUTE GLOBAL <string>result_var_name
    AS VOLUME INTEGRAL|SURFACE INTEGRAL
    OF NODAL|ELEMENT <string>source_var_name
```

```

COMPUTE GLOBAL <string>result_var_name
  AS SURFACE INTEGRAL OF ELEMENT ELEMENT_LOCAL_FORCE
  IN DIRECTION <real>dx <real>dy <real>dz
COMPUTE GLOBAL <string>result_var_name AS
  INTERNAL REACTION
  [AT POINT <real>px <real>py <real>pz]
  [IN DIRECTION <real>dx <real>dy <real>dz]
COMPUTE GLOBAL|ELEMENT <string>result_var_name AS INTERPOLATION
  OF NODAL <string>source_var_name
  AT POINT <real>x <real>y <real>z
  IN MODEL|CURRENT COORDINATES

```

```

COMPUTE GLOBAL|NODAL|ELEMENT|FACE <string>result_var_name
  AS FUNCTION <string>function_name
COMPUTE GLOBAL <string>result_var_name
  FROM EXPRESSION "<string>expression"
COMPUTE GLOBAL <string>result_var_name
  AS INTEGRAL OVER TIME|LOG|EXP|MAGNITUDE OF GLOBAL
  <string>source_var_name

```

```

COMPUTE NODAL <string>result_var_name AS
  MAX OVER TIME|MIN OVER TIME|ABSOLUTE VALUE MAX OVER
  TIME|AVERAGE OVER TIME|STANDARD DEVIATION OVER TIME|
  LOG|EXP OF NODAL <string>source_var_name
COMPUTE NODAL <string>result_var_name AS
  END STEP VALUE OF NODAL VELOCITY

```

```

COMPUTE ELEMENT <string>result_var_name AS
  MAX OVER TIME|MIN OVER TIME|ABSOLUTE VALUE MAX OVER TIME|
  AVERAGE OVER TIME|STANDARD DEVIATION OVER TIME
  OF ELEMENT <string>source_var_name
COMPUTE ELEMENT <string>result_var_name AS
  MAX|MIN|AVERAGE|LOG|EXP OF ELEMENT <string>source_var_name
COMPUTE ELEMENT|FACE <string>result_var_name AS
  MAX|MIN OF NODAL <string>source_var_name
COMPUTE ELEMENT|FACE <string>result_var_name AS
  AVERAGE OF NODAL <string>source_var_name
  [WEIGHTED BY <string>weight_var_name]

```

```

#
# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name |
  SURFACE SUBROUTINE = <string>subroutine_name |
  ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name

```

 **Explicit**

```

    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
#
# copy command
COPY ELEMENT VARIABLE <string>ev_name TO NODAL VARIABLE
    <string>nv_name
#
# extrapolate command
EXTRAPOLATE ELEMENT VARIABLE <string>ev_name to NODAL VARIABLE
    <string>nv_name
[ LINEAR SOLVER = <string>solver_name (DEFAULT_FETI_SOLVER)]
#
# variable transformation command
TRANSFORM NODAL|ELEMENT VARIABLE <string>variable_name
    TO COORDINATE SYSTEM <string>coord_sys_name
    AS <string> transformed_name
#
# data filtering
FILTER <string>new_var FROM NODAL|ELEMENT|FACE
    <string>source_var USING <string>filter_name
#
# compute for element death
COMPUTE AT EVERY STEP
#
# fine-grained control of when output is computed
COMPUTE AT HEARTBEAT|HISTORY|RESULTS OUTPUT STEPS
#
# additional command
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
END [USER OUTPUT]

```

 **Explicit**

The `USER OUTPUT` command block enables the user to generate specialized output information which is derived from analysis results such as element stresses, displacements, and velocities. `USER OUTPUT` post-processes analysis results *in situ*, as the code is running, to produce a reduced set of specialized output information. The `USER OUTPUT` command block offers an alternative to writing out large quantities of data and then post-processing them with an external code to produce customized output results.

As an example, the `USER OUTPUT` command block could be used to sum the contact forces in a particular global axis direction, on a certain surface in the mesh, to yield a net resultant contact force on that surface. In this instance, `USER OUTPUT` would reduce a potentially large set of detailed contact data to a single, meaningful value for a surface (or set of surfaces).



In an explicit analysis, the `USER OUTPUT` command block may also be used to define a user element failure criterion. In this case, the `COMPUTE AT EVERY STEP` command must be used to ensure that the variable is computed at every time step. See Section 9.3.2.8 on element death for additional details.

There are several options for calculating user-defined quantities:

Compute A single command line in the command block is used to compute reductions of variables on subsets of the mesh. This option makes use of one of the `COMPUTE` result command lines. For instance, the above example of the contact force is a case where we can accomplish the desired result simply by using the `COMPUTE GLOBAL` command line.

User subroutine The command block specifies a user subroutine to run immediately preceding output to calculate any desired variable. This option makes use of a `NODE SET`, `SURFACE`, or `ELEMENT BLOCK SUBROUTINE` command line.

Copy There is an option to copy an element variable for an element to the nodes associated with the element, via the `COPY ELEMENT VARIABLE` command line, made available primarily for creating results files for some of the post-processing tools used with Sierra/SM.

Only one of these three options may be used in a given `USER OUTPUT` command block.

For the compute result option, a user-defined variable is automatically generated. This user-defined variable is given whatever name the user selects for `results_var_name` in the above specification for any of the three compute result command lines. Parenthesis syntax (Section 9.1) may be used to define reductions on integration points or components of a variable. By default, a reduction operation operates on each integration point. For example, if the compute global command was used to average values of element stress it would average the values of stress at all integration points of all elements. If the command was used to average `stress(:, 1)`, the result would only be the average of stress on the first integration points.

If the user subroutine or copy option is used, the user will need to define some type of user variable with the `USER VARIABLE` command block described in Section 11.2.4.

User-defined variables, whether they are generated via the compute result option or the `USER VARIABLE` command block, are not automatically written to a results or history file. If the user wants to output any user-defined variables, these variables must be referenced in a results or history output specification (see Section 9.3.1 and Section 9.4, which describe the output of variables to results files and history files, respectively).

The `USER OUTPUT` command block contains five groups of commands—mesh-entity set, compute result, user subroutine, copy, and compute for element death. Each of these command groups is independent of the others. In addition to the command lines in the five command groups, there is an additional command line: `ACTIVE PERIODS`. The following sections provide descriptions of the command groups and the `ACTIVE PERIODS` command line.

9.3.2.1 Mesh-Entity Set Commands

The `mesh-entity set` commands portion of the `USER OUTPUT` command block specifies the nodes, element faces, or elements associated with the variable to be output. This portion of the

command block can include some combination of the following command lines:

```
NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list> surface_names
REMOVE BLOCK = <string list>block_names
```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes, element faces, or elements. See Section 7.1.1 for more information about the use of these command lines for mesh entities. If no mesh-entity commands are defined for user output, the output will be computed for all blocks in the mesh.

9.3.2.2 Compute Result Commands

The compute result commands are used to compute new variables by performing operations on an existing variable. Several general forms of the compute results command are supported shown below.

```
# Compute global norms or reductions of nodal, face, or
# element quantities
PROCESS ELEMENTS = ACTIVE_ONLY|INACTIVE_ONLY|ALL(ACTIVE_ONLY)
COMPUTE GLOBAL <string>result_var_name AS
    SUM|MAX|MIN|MAX ABSOLUTE VALUE|L2NORM|VARIANCE
    OF NODAL|FACE|ELEMENT <string>source_var_name
COMPUTE GLOBAL <string>result_var_name
    AS AVERAGE
    OF NODAL|ELEMENT|FACE <string>source_var_name
    [WEIGHTED BY <string>weight_var_name]
COMPUTE GLOBAL <string>result_var_name AS
    VOLUME INTEGRAL|SURFACE INTEGRAL
    OF NODAL|ELEMENT <string>source_var_name
COMPUTE GLOBAL <string>result_var_name
    AS SURFACE INTEGRAL OF ELEMENT ELEMENT_LOCAL_FORCE
    IN DIRECTION <real>dx <real>dy <real>dz
COMPUTE GLOBAL <string>result_var_name AS
    INTERNAL REACTION
    [AT POINT] <real>px <real>py <real>pz
    [IN DIRECTION] <real>dx <real>dy <real>dz
COMPUTE GLOBAL|ELEMENT <string>result_var_name AS INTERPOLATION
    OF NODAL <string>source_var_name
    AT POINT <real>x <real>y <real>z
    IN MODEL|CURRENT COORDINATES
```

The COMPUTE GLOBAL reduction result command computes a single global value or set of global values by performing a user-specified operation on the given nodal or element variable `source_`

var_name. A single global value, for example, might be the maximum value of one of the stress components of all the elements in our specified set, while a set of global values would be the maximum value of each stress component over all elements in the specified set.

The `PROCESS ELEMENTS` command line may be optionally specified. By default, global reductions are performed only on the active element set, which excludes elements killed by element death or other processes. To consider all elements, only alive elements, or only dead elements, options `ALL`, `ACTIVE_ONLY`, or `INACTIVE_ONLY`, respectively, may be specified.

```
# Store current value of an XY function into a global quantity
COMPUTE GLOBAL|NODAL|ELEMENT|FACE <string>result_var_name
    AS FUNCTION <string>function_name

# Store current value of a scalar expression into a global quantity
COMPUTE GLOBAL <string>result_var_name
    FROM EXPRESSION "<string>expression"

# Compute a global quantity from the time history of an existing
# global quantity
COMPUTE GLOBAL <string>result_var_name
    AS INTEGRAL OVER TIME OF GLOBAL <string>source_var_name

# Compute a global quantity from another global quantity
COMPUTE GLOBAL <string>result_var_name
    AS LOG|EXP|MAGNITUDE OF GLOBAL <string>source_var_name

# Compute an element quantity by operating on quantities stored
# at the element integration points
COMPUTE ELEMENT <string>result_var_name AS
    MAX|MIN|AVERAGE|SUM|LOG|EXP OF ELEMENT
    <string>source_var_name

# Compute an element or face quantity by operating on quantities
# stored at nodes attached to the element or face
COMPUTE ELEMENT|FACE <string>result_var_name AS
    MAX|MIN|SUM OF NODAL <string>source_var_name
COMPUTE ELEMENT|FACE <string>result_var_name AS
    AVERAGE OF NODAL <string>source_var_name
    [WEIGHTED BY <string>weight_var_name]

# Compute a nodal quantity from the time history of an existing
# nodal quantity
COMPUTE NODAL <string>result_var_name AS
    MAX OVER TIME|MIN OVER TIME|ABSOLUTE VALUE MAX OVER TIME|
    AVERAGE OVER TIME|STANDARD DEVIATION OVER TIME
    OF NODAL <string>source_var_name
```




```
COMPUTE NODAL <string>result_var_name AS
    END STEP VALUE OF NODAL VELOCITY

# Compute a nodal quantity from another nodal quantity
COMPUTE NODAL <string>result_var_name AS
    LOG|EXP OF NODAL <string>source_var_name

# Compute an element quantity from the time history of an existing
# element quantity
COMPUTE ELEMENT <string>result_var_name AS
    MAX OVER TIME|MIN OVER TIME|ABSOLUTE VALUE MAX OVER TIME|
    AVERAGE OVER TIME|STANDARD DEVIATION OVER TIME
    OF ELEMENT <string>source_var_name
```

The `COMPUTE NODAL` and `COMPUTE ELEMENT` variable commands compute a new nodal or element field or user variable by operating on an existing field, for example, to compute the maximum stress in each element over the course of an analysis.

The `AS FUNCTION` option uses a time history function `function_name` to compute a global, element, node, or face variable. The function can be either a simple time history function or a more complex analytic function with custom arguments. In the analytic function, a `GLOBAL` variable may only be defined as a function of other `GLOBAL` variables. `NODAL`, `ELEMENT`, and `FACE` variables may be a function of `GLOBAL` variables and other `NODAL`, `ELEMENT`, or `FACE` variables, respectively; for instance, face variables may not depend on node variables.

The `FROM EXPRESSION` option computes a new global variable by evaluating an expression involving other global variables. For example

```
COMPUTE GLOBAL energyNorm FROM EXPRESSION
    "max(abs(KineticEnergy), abs(InternalEnergy))"
```

would compute a new global variable called `energyNorm`, which is the maximum absolute value of either kinetic or internal energy at a given time.

The `INTEGRAL OVER TIME` option computes a new global variable by computing the time integral of an existing global variable from the analysis start time up until the current time.

In the above command lines, the following definitions apply:

- The string `source_var_name` is the name of the variable used to compute the result variable. (See Section 9.10 for a listing of available code variables).
- The string `result_var_name` is the name of a new variable in which the computed results are to be stored. To output its value, `result_var_name` may be specified in `RESULTS`, `HEARTBEAT`, and/or `HISTORY OUTPUT` blocks. The result variable may also be used as a source variable in other `USER OUTPUT` command blocks.

- The `SUM`, `AVERAGE`, `MAX`, `MIN`, and `MAX ABSOLUTE VALUE` global reduction commands compute the sum, average, maximum value, minimum value, and maximum absolute value of the given field, respectively, over all included mesh entities. The `AVERAGE` command may also take an optional `WEIGHTED BY` field to compute a weighted average. The weighted by field may be given any scalar field defined on the same entities as the source variable. Mostly commonly this weighted by field would be used to compute a mass or volume weighted average value.
- The `L2NORM` global reduction computes the vector L_2 -norm, or Euclidean norm, of the given field on all included entities. The vector L_2 -norm computes the square root of the sum of the squares of all field values. It is important to note that this output command does *not* compute a volume-integrated L_2 -norm.
- The `VARIANCE` global reduction computes the variance over all included mesh entities, defined as the average of the squared differences from the mean.
- The `LOG` and `EXP` commands calculate the natural logarithm and exponential, respectively, of the variable on all included mesh entities. Logarithm and exponential are valid for scalars, symmetric tensors, and full tensors. A logarithmic or exponential mapping, respectively, is performed on symmetric and full tensors rather than a component-wise scalar logarithm or exponential.



Warning: The `LOG` command is only defined for non-zero scalars and certain types of tensors (see [5] for details).

- The `MAGNITUDE` command calculates the magnitude of a global variable, computed as the square root of the sum of the squares of the variable components. For a scalar value the magnitude is the absolute value, and for a vector value it is the vector length.
- The `INTERNAL REACTION` command is a special user output reduction that computes the internal reaction on an interior plane of the model. To use this command a set of elements or blocks must be specified in the user output command block. The options `AT POINT` and `IN DIRECTION` optionally define a point on the plane and the plane normal, respectively, given in initial model coordinates; if not specified, by default the body's center of mass and the eigenvector corresponding to the minimum eigenvalue of the inertia tensor are used.

The reaction is computed as the net internal force across the specified plane. At equilibrium the internal force pulling on each side of the plane should be equal and opposite. If the body is not in equilibrium, however, the internal forces on either side of the plane may not be equal, indicating a net acceleration on the plane. In this case the value computed is $0.5(\mathbf{F}^+ - \mathbf{F}^-)$, where $\mathbf{F}^+$ and $\mathbf{F}^-$ are the forces on the positive and negative sides of the plane, respectively.

- The `AS INTERPOLATION` command can be used in two different ways. In the first case, nodal variables can be interpolated to a specific point which stores the value in a global variable. The interpolation point coordinates can be defined in either initial model coordinates or current coordinates. The interpolation is performed using the containing element's shape

functions, so that the point moves with the deformation of the model in a Lagrangian manner. Eight-noded hexahedral and four-noded linear and ten-noded quadratic tetrahedral solid elements are currently supported. The analysis will fail if a point outside of the model is given; the log file will describe the error, including the point that caused it.

In the second case, nodal variables can be interpolated to an element's integration points. Again, the interpolation is performed using the element's shape functions, unless the field being interpolated has fewer values than the element has integration points. If this is the case, the average is computed for the element. Eight-noded hexahedral, four-noded linear tetrahedral, and ten-noded quadratic tetrahedral solid elements are currently supported.

To interpolate an element variable at a point, it must first be projected to a nodal variable via the `EXTRAPOLATE` user output command. The projected nodal variable may then be interpolated to a point as usual. It is important to note that the `EXTRAPOLATE` command will yield a continuous projected nodal field across element block boundaries; to circumvent this limitation for inherently discontinuous element fields at block boundaries, such as stress at material interfaces, separate `EXTRAPOLATE` commands should be provided per element block, and the projected block-specific nodal variables may then be interpolated to points of interest. For example,

```
BEGIN USER OUTPUT
  BLOCK = block_1
  EXTRAPOLATE ELEMENT VARIABLE stress(xx) TO
    NODAL VARIABLE stress_b1
END USER OUTPUT

BEGIN USER OUTPUT
  BLOCK = block_2
  EXTRAPOLATE ELEMENT VARIABLE stress(xx) TO
    NODAL VARIABLE stress_b2
END USER OUTPUT

BEGIN USER OUTPUT
  INCLUDE ALL BLOCKS
  # interpolate to a point inside block_1
  COMPUTE GLOBAL stress_pt1 AS INTERPOLATION OF
    NODAL stress_b1 AT POINT 0.0 0.0 0.0
  # interpolate to a point inside block_2
  COMPUTE GLOBAL stress_pt2 AS INTERPOLATION OF
    NODAL stress_b2 AT POINT 5.0 0.0 0.0
END USER OUTPUT
```

For additional guidance on the `EXTRAPOLATE` user output command, consult [Section 9.3.2.5](#).

- The `VOLUME INTEGRAL` global reduction is valid for nodal or element fields. The volume integral of an element field is computed as $\sum_{i=1}^N f_i w_i$ where N is the number of quadrature points, f_i are the field values at the integration points, and w_i are the effective quadrature point volumes. If a uniform gradient element is used, $N = 1$, and w_1 represents the element volume. The volume integral of a nodal field is computed in the same manner, but nodal

field values must first be interpolated to the quadrature points of each element using the element shape functions. The quadrature points for integration of a nodal field are always taken from the fully integrated element, even if a uniform gradient element is used in the analysis; this method improves integration accuracy and is not likely to affect performance. Volume weighting is based on the reference (initial) quadrature point volumes and not the deformed mesh volumes.

- The `SURFACE INTEGRAL` global reduction is nearly identical to the `VOLUME INTEGRAL` option. However, in this case, integration is only performed over surfaces of a given element, so a side set must be specified in the containing user output command block.

Generic surface integration currently only supports variables of `NODAL` type; however, `SURFACE INTEGRAL` may also be used as a specialized user output for computing the net internal force over a sideset. The command operates on the internal force on the side set in the specified direction, yielding the net internal force magnitude in that direction. Note that this capability is only compatible with the element variable `element_local_force`.

- Five different methods are available for computing nodal or element variables over all time steps in the analysis: `MAX OVER TIME`, `MIN OVER TIME`, `ABSOLUTE VALUE MAX OVER TIME`, `AVERAGE OVER TIME`, or `STANDARD DEVIATION OVER TIME` which respectively compute the maximum, minimum, absolute maximum, average, or standard deviation of values in the source variable over all time.
-  **Explicit** The `AS END STEP VALUE OF` command gives the user the option to output the end-step velocity. In the central difference time integrator implementation, velocities are typically stored at the half-step; i.e., at a given time t_n , the default velocity output yields values at $t_n - 0.5\Delta t_n$ rather than t_n . This user output option moves the velocities to t_n and stores them in `result_var_name`.

Usage guidelines for the above `COMPUTE` commands are as follows:

- If both the source variable and target variable are element variables, and `MAX`, `MIN`, or `AVERAGE` is used, then the operation computes a single element variable based on a reduction of the values at all integration points for that element. `AVERAGE` returns the volumetric weighted average over the element.
- Note that global reduction methods such `SUM`, `AVERAGE`, `MAX`, `MIN`, and `MAX ABSOLUTE VALUE` are only performed on active nodes and elements; this behavior can be changed with the `PROCESS ELEMENTS` command.
- The source variable used to compute a global variable must be either a nodal quantity or an element quantity, as specified by the `NODAL` or `ELEMENT` option.
- The source variable used to compute a nodal or element variable must be of the same type as the result variable.
- Standard component syntax may also be used in the `source_var_name` to specify operations on sub-components of a given variable (e.g., `stress(xx)`).

Examples Compute the total x -direction reaction force on `surface_1`:

```
SURFACE = surface_1
COMPUTE GLOBAL wall_x_reaction AS SUM OF NODAL reaction(x)
```

Compute the maximum force seen by each spot weld at the spot weld connection node.

```
COMPUTE NODAL max_spotn AS MAXIMUM OVER TIME OF NODAL #
    spot_weld_normal_force
```

Compute the volumetric average value of `eqps` in an element with multiple integration points. This is the sum of integration point values weighted by the relative integration point volumes.

```
COMPUTE ELEMENT eqps_avg AS AVERAGE OF ELEMENT eqps
```

Compute the approximate element centroid as the average the element coordinates:

```
COMPUTE ELEMENT elem_cent AS AVERAGE OF NODAL coordinates
```

Compute the total mass of elements removed by element death or other element removal processes.

```
COMPUTE GLOBAL removedMass AS SUM OF ELEMENT element_mass
PROCESS ELEMENTS = INACTIVE_ONLY
```

9.3.2.3 User Subroutine Commands

If the user subroutine option is invoked, the user-defined output quantities will be calculated by a subroutine written by the user explicitly for this purpose. The subroutine will be called by Sierra/SM at the appropriate time to perform the calculations. User subroutines allow for more generality in computing user-defined results than the `COMPUTE GLOBAL` command line. A subroutine could be used, for example, in case an analytic solution exists for a given problem, and it is desired to compute the difference between the analytic value and a corresponding computed value throughout an analysis. A full detailed description of user subroutines is given in [Chapter 11](#).

The following command lines are related to the user subroutine option:

```
NODE SET SUBROUTINE = <string>subroutine_name |
    SURFACE SUBROUTINE = <string>subroutine_name |
    ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

The user subroutine option is invoked by using the `NODE SET SUBROUTINE` command line, the `SURFACE SUBROUTINE` command line, or the `ELEMENT BLOCK SUBROUTINE` command line. The particular command line selected depends on the mesh-entity variable type for which the result quantities are being calculated. For example, variables associated with nodes would be calculated by using a `NODE SET SUBROUTINE` command line, variables associated with faces by using a `SURFACE SUBROUTINE` command line, and variables associated with elements by using the `ELEMENT BLOCK SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine written by the user. A user subroutine in the `USER OUTPUT` command block returns no values. Instead, it performs its operations directly with commands such as `aupst_put_nodal_var`, `aupst_put_elem_var`, and `aupst_put_global_var`. See Chapter 11 for further discussion of these various put commands.

Following the selected command line (`NODE SET SUBROUTINE`, `SURFACE SUBROUTINE`, or `ELEMENT BLOCK SUBROUTINE`) are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and consist of `SUBROUTINE DEBUGGING OFF`, `SUBROUTINE DEBUGGING ON`, `SUBROUTINE REAL PARAMETER`, `SUBROUTINE INTEGER PARAMETER`, and `SUBROUTINE STRING PARAMETER`. Examples of using these command lines are provided in Chapter 11.

Importantly, to implement the user subroutine option and output the calculated information, it is necessary to

1. create the user-defined variable with a `USER VARIABLE` command block;
2. calculate the results for the user-defined variable in the user subroutine; and
3. write the results for the user-defined variable to an output file by referencing it in a `RESULTS OUTPUT` command block and/or a `HISTORY OUTPUT` command block and/or a `HEARTBEAT OUTPUT` command block. In the `RESULTS OUTPUT` command block, a `NODAL`, `ELEMENT`, or `GLOBAL` command line would be used, depending on the `USER VARIABLE` command block definition. Similarly, in the `HISTORY OUTPUT` or `HEARTBEAT OUTPUT` command block, the appropriate form of the variable command line should be used.

9.3.2.4 Copy Command

```
COPY ELEMENT VARIABLE <string>ev_name TO NODAL VARIABLE
<string>nv_name
```

The `COPY ELEMENT VARIABLE` command line copies the value of an element variable to a node associated with the element. The element variable to be copied is specified by `ev_name`; the name of the nodal variable to which the value is being transferred is `nv_name`. The nodal variable must be specified as a user-defined variable.

9.3.2.5 Extrapolate Command

```
EXTRAPOLATE ELEMENT VARIABLE <string>ev_name TO NODAL VARIABLE
<string>nv_name
```

The `EXTRAPOLATE ELEMENT VARIABLE` command line projects element variables to the nodes via L_2 projection. The element variable to be extrapolated is specified by `ev_name`. The nodal variable to which the element variable is projected is a user-defined variable denoted `nv_name`.

The L_2 projection operates on the inverse of the consistent mass matrix corresponding to the specified block(s). The extrapolation option is available for most solid elements such as hexahedra, tetrahedra, and quadrilateral cohesive zone elements.



Warning: Because the extrapolation operates on element variables, the `NODE SET =` command line is *not* compatible with user output extrapolation operations; either specific element blocks or `INCLUDE ALL BLOCKS` must be specified.

L_2 projection using the uniform gradient, fully integrated, and composite tetrahedral formulations of the 10-noded tetrahedron yields spurious nodal values at the 6 edge nodes. Instead, each 10-noded tetrahedron is converted to an equivalent 4-noded, linear tetrahedron based at the corner nodes during the projection phase. The extrapolated corner node values are then interpolated to the higher-order edge nodes of the original 10-noded tetrahedron.

The extrapolation can be restricted to specific element blocks with the `BLOCK =` command line. A *global* extrapolation linear system will be shared among all user output command blocks in which `INCLUDE ALL BLOCKS` is specified. However, a separate *local* extrapolation linear system will be created for each user output command block specifying individual blocks.



Warning: Global extrapolation should be avoided on heterogeneous sets of elements such as combinations of hexahedra and cohesive zones due to inconsistencies in their respective topological formulations.



Warning: Specifying the same nodal variable name for multiple element blocks and/or multiple user output extrapolation operations may lead to inconsistencies in nodal extrapolated quantities at any shared block boundaries. When extrapolating the same element variable (e.g., `stress`) to nodes of multiple blocks, it is recommended to specify a unique nodal output variable name on each block (e.g., `nodal_stress_block_1`, `nodal_stress_block_2`, etc.).

A particular linear solver may be selected for the L_2 projection using the `LINEAR SOLVER =` user output line command. The default is `DEFAULT_FETI_SOLVER`.

9.3.2.6 Variable Transformation Command

```
TRANSFORM NODAL|ELEMENT VARIABLE <string>variable_name
  TO COORDINATE SYSTEM <string>coord_sys_name
  AS <string> transformed_name
```

The `TRANSFORM NODAL|ELEMENT VARIABLE` command line transforms a nodal vector (dis-

placement, velocity, acceleration, etc) or an element tensor (stress, strain, etc.) from components in the global coordinate system to components in the coordinate system defined in a `BEGIN COORDINATE SYSTEM` command block having the name `coord_sys_name` (see Section 2.1.9). The transformed variables will be output to the results file as `transformed_name`.

By default most vectors and tensors are rotated into a new local $X'Y'Z'$ Cartesian coordinate system. As an example consider transforming a nodal force to a cylindrical coordinate system. A local Cartesian $X'Y'Z'$ coordinate system will be computed at the node location where X' aligns with the cylindrical radial direction, Y' aligns with the cylindrical hoop direction, and Z' aligns with the cylindrical Z direction. The force will then be transformed into this new Cartesian coordinate system in a way that is length preserving as shown in Figure 9.1.

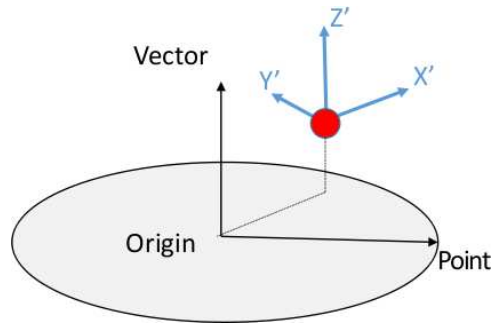


Figure 9.1: Default Cylindrical Transformation

The nodal fields `coordinates` and `model_coordinates` are treated specially. These positional vectors are transformed to new positions in the target coordinate system. This special position transformation only applies to these two specific nodal field names. For cylindrical coordinate system the coordinate vectors are transformed into $R \ \Theta \ Z$ coordinates where Θ is in radians as shown in Figure 9.2.

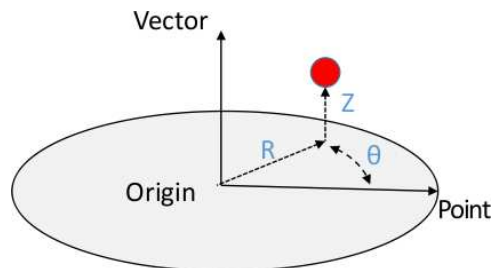


Figure 9.2: Cylindrical System Transformation of Coordinates



Warning: This command cannot be used to transform shell element tensors which are not computed in the global coordinate system. Output of shell stress and strain components in a user-defined, local co-rotational coordinate system are obtained with the element variables `transform_shell_stress` and `transform_shell_strain` as described in Section 9.3.1.



Warning: This command cannot be used with derived output variables. The derived output variables are variables such as `log_strain`, `biot_strain` and `green_lagrange_strain` which are not internally stored but are computed before being output to the results file

9.3.2.7 Data Filtering Commands

```
BEGIN FILTER <string>filter_name
  ACOEFF = <real_list>a_coeff
  BCOEFF = <real_list>b_coeff
  INTERPOLATION TIME STEP = <real>ts
END [FILTER]

BEGIN USER OUTPUT
  FILTER <string>new_var FROM NODAL|ELEMENT|FACE
    <string>source_var USING <string>filter_name
END
```

The user output `FILTER` command creates a new variable `new_var` by performing an on-the-fly frequency filter of the named element, nodal or face variable `source_var`. The `BEGIN FILTER` command block defines a filter with the name `filter_name`. The filter defined by the begin filter command block is then referenced by the `USER OUTPUT` filter command.

In the `BEGIN FILTER` command block the A and B filtering coefficients are defined with the `ACOEFF` and `BCOEFF` command lines. The filter must define at least one A and one B coefficient, there is no maximum number coefficients and the length of A and B do not need to match.

Filter operations assume that the data given at a constant time step. The explicit dynamics time step will tend to vary during the analysis. The `INTERPOLATION TIME STEP` command is used to linearly interpolate the data being produced at a non-constant time step down to some specified constant time step. The interpolation time step must be larger than zero and ideally should be specified to be smaller than the smallest time step with which the computations will iterate.

One way to obtain the filtering coefficients is with MATLAB. The following is an example of defining a third order Butterworth filter with a pass frequency of 100Hz at data interpolated to a time step of $1.0e - 5$ seconds. The filter is then used to filter acceleration histories of the nodes to 100Hz. The MATLAB code below will give the desired filtering coefficients. The full 16-digit precision of the coefficients returned by MATLAB should be used. If truncated precision numbers are used, the filters can potentially be unstable.

```
clear;
format long e;
passFrequency = 100;
interp_ts = 1.0e-5;
butterCoeff = 2.0*interp_ts*passFrequency;
[bcoeff,acoeff] = butter(3,butterCoeff);
```

```
acoeff
bcoeff
```

The computed filtering coefficients can be used in a `USER OUTPUT` block as show below. If the analysis time step always remains above $1.0e - 5$ the filter will be valid. If the analysis time step drops below $1.0e - 5$ there could be aliasing issues, and a smaller interpolation time step should be specified.

```
BEGIN FILTER filt_100Hz
  ACOEFF = 1.000000000000000e+00 -2.987433650055722e+00 $
          2.974946132665442e+00 -9.875122361107358e-01
  BCOEFF = 3.081237301416628e-08 9.243711904249885e-08 $
          9.243711904249885e-08 3.081237301416628e-08
  INTERPOLATION TIME STEP = 1.0e-5
END
BEGIN USER OUTPUT
  FILTER ax100Hz FROM NODAL acceleration(x) USING filt_100Hz
  FILTER ay100Hz FROM NODAL acceleration(y) USING filt_100Hz
  FILTER az100Hz FROM NODAL acceleration(z) USING filt_100Hz
END
```

9.3.2.8 Compute at Every Step Command

```
COMPUTE AT EVERY STEP
```

If this command line appears in the `USER OUTPUT` command block, a user-defined variable in the command block will be written at every time step. Section 11.2.4 discusses user-defined variables. In explicit dynamics analyses, user-defined variables can be used as criteria for element death. In this case, the `COMPUTE AT EVERY STEP` command must be used to force those variables to be computed at every time step. See Section 6.5 for more information on element death.



Explicit Only

9.3.2.9 Additional Commands

Active and Inactive Periods Commands

```
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The `ACTIVE PERIODS` and `INACTIVE PERIODS` command lines determine when the `USER OUTPUT` command block is either active or inactive. See Section 2.6 for more information about this optional command line.

Compute at Output Steps Commands

```
COMPUTE AT HEARTBEAT|HISTORY|RESULTS OUTPUT STEPS
```

This command line specifies to compute the current user output block at only heartbeat, history, or results output steps. For instance, if a particular `USER OUTPUT` command block is computationally intensive, but the results of the user output computation are only needed at relatively infrequent results output intervals, specifying `COMPUTE AT RESULTS OUTPUT STEPS` would exclude the user output block from being computed at history and heartbeat output steps and could result in a significant simulation runtime savings.

If this command line is not specified anywhere in a `USER OUTPUT` command block, the block will be computed at all output steps. It is admissible to specify multiple `COMPUTE AT OUTPUT STEPS` command lines per `USER OUTPUT` command block, for instance, if it is desired to compute at both history and heartbeat output steps.

9.4 History Output

```
BEGIN HISTORY OUTPUT <string>history_name
#
# Database Commands
DATABASE NAME = <string>history_file_name
DATABASE TYPE = <string>database_type(exodusII)
OVERWRITE = <string>OFF|ON|TRUE|FALSE|YES|NO
            (ON|TRUE|YES)
TITLE <string>user_title
PROPERTY = <string>propName = <string>propValue
#
# for global variables
GLOBAL <string>variable_name
    [AS <string>history_variable_name]
#
# for mesh entity - node, edge, face,
# element - variables
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name
    AT NODE|NODAL|EDGE|FACE|ELEMENT <integer>entity_id
    AS <string>history_variable_name
# for edge or face variables
EDGE|FACE <string>variable_name
    AT EDGE|FACE <integer>entity_id.local_face
    AS <string>history_variable_name
#
# for nearest point output of mesh entity - node,
# edge, face, element - variables
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name
    NEAREST LOCATION <real>global_x,
        <real>global_y>, <real>global_z
    AS <string>history_variable_name
START TIME = <real>output_start_time
TIMESTEP ADJUSTMENT INTERVAL = <integer>steps
AT TIME <real>time_begin INCREMENT =
    <real>time_increment_dt
ADDITIONAL TIMES = <real>output_time1
    <real>output_time2 ...
AT STEP <integer>step_begin INCREMENT =
    <integer>step_increment
ADDITIONAL STEPS = <integer>output_step1
    <integer>output_step2 ...
TERMINATION TIME = <real>termination_time_value
SYNCHRONIZE_OUTPUT
USE OUTPUT SCHEDULER <string>scheduler_name
OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|
    SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|
    SIGKILL|SIGILL|SIGSEGV
```

 **Explicit**

```
END [HISTORY OUTPUT <string>history_name]
```

A history file gives nodal variable results (displacements, forces, etc.) for specific nodes, edge variable results for specific edges, face variable results for specific faces, element results (stress, strain, etc.) for specific elements, and global results at specified times. A history file, the results to be included in this file, and the frequency at which results are written may be specified by a `HISTORY OUTPUT` command block. The command block appears inside the region scope. Typically, node, element, and occasionally face variables may be included.

More than one history file can be specified for an analysis. For each history file will correspond to one `HISTORY OUTPUT` command block in the input file. The command block for a history file description begins with:

```
BEGIN HISTORY OUTPUT <string>history_name
```

and is terminated with:

```
END [HISTORY OUTPUT <string>history_name]
```

where `history_name` is a user-selected name for the command block. Nested within the `HISTORY OUTPUT` command block are a set of command lines, as shown in the block summary given above.

See Section 9.2 for details on the general database commands such as `DATABASE NAME`, `DATABASE TYPE`, `OVERWRITE`, etc.

The other command lines that appear in the `HISTORY OUTPUT` command block determine the type and frequency of output information. Descriptions of these command lines follow in Section 9.4.1 through Section 9.4.5. The command lines for controlling the frequency of history output (in Section 9.4.1 through Section 9.4.5) are the same as those for controlling the frequency of results output. These frequency-related command lines are repeated here for convenience.

9.4.1 Output Variables

The `GLOBAL`, `NODE` (or `NODAL`), `EDGE`, `FACE`, or `ELEMENT` command line is used to select variables for output in the history file. One of several types of variables can be selected for output. The form of the command line varies depending on the variable type selected for output.

9.4.1.1 Global Output Variables

```
GLOBAL <string>variable_name  
[AS <string>history_variable_name]
```

This form of the command line lets the user select any global variable for output in the history file. The variable is selected with the string `variable_name`. The string `variable_name` is the name of the global variable and can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4).

A name, `history_variable_name`, can also be specified for the selected entity following the `AS` keyword. For example, suppose it is desired to output the CPU time (`cpu_time`) as `cpu` in the history file. The command line to accomplish this would be:

```
GLOBAL cpu_time AS cpu
```

The specification of an alias is optional for output of a global variable.

9.4.1.2 Mesh Entity Output Variables

```
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name
  AT NODE|NODAL|EDGE|FACE|ELEMENT <integer>entity_id
  AS <string>history_variable_name
```

This form of the command line allows the user to select any nodal, edge, face, or element variable for a specific mesh entity for output in the history file. For example, this form of the command line will pick the displacement at a specific node and output the displacement to the history file using a specified alias.

For this form of the command line, the mesh entity type is set to `NODE` (or `NODAL`), `EDGE`, `FACE`, or `ELEMENT` depending on the variable (set by `variable_name`) to be output. If the mesh entity type is set to `NODE` (or `NODAL`), `EDGE`, or `FACE`, the string `variable_name` can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4). If the mesh entity type is set to `ELEMENT`, the string `variable_name` can be a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4).

Selection of a mesh entity follows the `AT` keyword. A mesh entity type (`NODE` [or `NODAL`], `EDGE`, `FACE`, or `ELEMENT`) must be specified, followed by the integer identifier, `entity_id`, for the mesh entity. A name, `history_variable_name`, for the selected entity must also be specified following the `AS` keyword. For example, output of accelerations at node 88 would be specified using the command line:

```
NODE ACCELERATION AT NODE 88 AS accel_88
```

where `accel_88` is the name used for this history variable in the history file.

For face and edge variables, an additional method of specifying the face or edge entity is provided. This method selects the face or edge by specifying the element that the face or edge is a member of and the local face or edge number on that element. To use this method, the element id and the face ordinal are specified as `id.ordinal`. The numbering of an elements faces and edges is specified in the ExodusII users manual [1]. Note that the selected face or edge must exist in the mesh database as a face or edge on a sideset, or the face or edge must be created internally by the application.

As an example, the command line to obtain the pressure for the third face of element 42 for the history file would be:

```
FACE PRESSURE AT FACE 42.3 AS PRES_42.3
```

Either the keyword `NODE` or `NODAL` can be used for nodal quantities.

As an example, the command line to obtain the von Mises stress for element 1024 for the history file would be:

```
ELEMENT VON_MISES AT ELEMENT 1024 AS GM_1024
```

Material state variables may be output in the same way. If an element has the state variable ‘EQPS’ then the value of ‘EQPS’ could be obtained on the element with a statement of the form:

```
ELEMENT EQPS AT ELEMENT 1024 AS eqps_1024
```

Additionally, the at element command supports the standard variable component syntax. For example:

```
ELEMENT STRESS(XX) AT ELEMENT 1024 AS S_XX_1024
```

where S_XX_1024 is the name that will be used for this history variable in the history file.

9.4.1.3 Nearest Point Output Variables

```
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name  
  NEAREST LOCATION <real>global_x,  
    real<global_y>, real<global_z>  
  AS <string>history_variable_name
```

This form of the command line allows the user to select any nodal, edge, face, or element variable for output in the history file using a nearest point criterion. The command line described in this subsection is an alternative to the command line described in the preceding section, Section 9.4.1.2, for obtaining history output. The command line in this section or the command line in Section 9.4.1.2 produces history files with variable information. The difference between the two is simply in how the variable information is selected.

For the above form of the command line, the mesh entity type is set to NODE (or NODAL), EDGE, FACE, or ELEMENT depending on the variable (set by `variable_name`) to be output. If the mesh entity type is set to NODE (or NODAL), EDGE, or FACE, the string `variable_name` can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4). If the mesh entity type is set to ELEMENT, the string `variable_name` can be a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4).

The specific mesh entity used for output is determined by global coordinates specified by the NEAREST LOCATION keyword and its associated input parameters—`global_x`, `global_y`, `global_z`. The specific mesh entity chosen for output is as follows:

- If the mesh entity has been set to NODE (or NODAL), the node in the mesh selected for output is the node whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.
- If the mesh entity has been set to EDGE, the edge in the mesh selected for output is the edge with a center point (the average location of the two end points of the edge) whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.
- If the mesh entity has been set to FACE, the face in the mesh selected for output is the face with a centroid whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.
- If the mesh entity has been set to ELEMENT, the element in the mesh selected for output is the element with a centroid whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.

In all the above cases, the original model coordinates are used when selecting the nearest entity, not the current coordinates.

A name, `history_variable_name`, must be specified for the selected entity following the `AS` keyword. As an example, to output the accelerations at a node closest to the point with global coordinates (1012.0, 54.86, 103.3141), the command line

```
NODE ACCELERATION
      NEAREST LOCATION 1012.0, 54.86, 103.3141 AS accel_near
```

would be used, where `accel_near` is the name used for this history variable in the history file. Either the keyword `NODE` or `NODAL` can be used for nodal quantities.

9.4.2 Outputting History Data on a Node Set

It is commonly desired to output history data on a single-node node set. If a mesh file is slightly modified, the node and element numbers will completely change. The node associated with a node set, however, remains the same, i.e., the node in the node set retains the same initial geometric location with the same connectivity to other elements, even when its node number changes. Therefore, we might want to specify the history output for a node set with a single node rather than with the global identifier for a node. This can easily be accomplished, as follows:

```
begin user output
  node set = nodelist_1
  compute global disp_ns_1 as average of nodal displacement
end

begin history output
  global disp_ns_1
end
```

If `nodelist_1` contains only a single node, the history output variable `disp_ns_1` will contain the displacement for the single node in the node set. If `nodelist_1` contains multiple nodes, the average displacement of the nodes will be output.

9.4.2.1 Specifying Time Steps for Output

All of the same output step control commands available for results output are also available for history output. See Section [9.3.1.9](#).

9.4.3 Synchronize Output

```
SYNCHRONIZE OUTPUT
```

The `SYNCHRONIZE OUTPUT` command can be used in the same manner it is used in the `exodus` output command block. See Section [9.3.1.10](#) for more details about this command.

9.4.4 Use Output Scheduler

```
USE OUTPUT SCHEDULER <string>scheduler_name
```

The `USE OUTPUT SCHEDULER` command can be used in the same manner it is used in the `exodus` output command block. See Section 9.3.1.11 for more details about this command.

9.4.5 Write History If System Error Encountered

```
OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|  
SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|  
SIGKILL|SIGILL|SIGSEGV
```

The `OUTPUT ON SIGNAL` command line is used to initiate the writing of a history file when the system encounters a type of system error. Only one error type in the list of error types should be entered for this command line. These system errors cause the code to terminate before the code can add any current history output (history output past the last history output time step) to the history file. If the code encounters the specified error type during execution, a history file will be written before execution is terminated. This command line can also be used to force the writing of a history file at some point during execution of the code. Suppose the command line

```
OUTPUT ON SIGNAL = SIGUSR2
```

is included in the input file. While the code is running, a user can execute (from the keyboard) the system command line

```
kill -s SIGUSR2 pid
```

to terminate execution and force the writing of a results file. In the above system command line, *pid* is the process identifier, an integer. The `OUTPUT ON SIGNAL` command line is primarily a debugging tool for code developers.

9.5 Heartbeat Output

```
BEGIN HEARTBEAT OUTPUT <string>heartbeat_name
# Can also use predefined streams "cout", "stdout",
# "cerr", "clog", "log", "output", or "outputP0"
STREAM NAME = <string>heartbeat_file_name
APPEND = ON|OFF (OFF)
#
# Specify whether heartbeat file will be in spyhis (cth)
# format, or default format
FORMAT = SPYHIS|ORIGINAL|CSV(CSV)
#
# for global variables
GLOBAL <string>variable_name
    [AS <string>heartbeat_variable_name]
#
# for mesh entity - node, edge, face,
# element - variables
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name
    AT NODE|NODAL|EDGE|FACE|ELEMENT <integer>entity_id
    AS <string>heartbeat_variable_name
#
# for nearest point output of mesh entity - node,
# edge, face, element - variables
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name
    NEAREST LOCATION <real>global_x,
    <real>global_y>, <real>global_z
    AS <string>heartbeat_variable_name
# for edge or face variables
EDGE|FACE <string>variable_name
    AT EDGE|FACE <integer>entity_id.local_face
    AS <string>heartbeat_variable_name
START TIME = <real>output_start_time
TIMESTEP ADJUSTMENT INTERVAL = <integer>steps
AT TIME <real>time_begin INCREMENT =
    <real>time_increment_dt
ADDITIONAL TIMES = <real>output_time1
    <real>output_time2 ...
AT STEP <integer>step_begin INCREMENT =
    <integer>step_increment
ADDITIONAL STEPS = <integer>output_step1
    <integer>output_step2 ...
TERMINATION TIME = <real>termination_time_value
SYNCHRONIZE_OUTPUT
USE OUTPUT SCHEDULER <string>scheduler_name
OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|
    SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|
    SIGKILL|SIGILL|SIGSEGV
```

 **Explicit**

```

PRECISION = <integer>precision
LABELS = <string>OFF|ON
LEGEND = <string>OFF|ON
TIMESTAMP FORMAT <string>timestamp_format
MONITOR = <string>RESULTS|RESTART|HISTORY

END [HEARTBEAT OUTPUT <string>heartbeat_name]

```

The heartbeat output is text output file that gives:

- nodal variable results (displacements, forces, etc.) for specific nodes,
- edge variable results for specific edges,
- face variable results for specific faces,
- element results (stress, strain, etc.) for specific elements, and
- global results

at specified times.



Known Issue: user-defined variables (see Section [11.2.4](#)) are not currently supported with heartbeat output.

The output is written as text instead of the binary history output. A heartbeat file, the results to be included in this file, the formatting of the output, and the frequency at which results are written may be specified using a `HEARTBEAT OUTPUT` command block. The command block appears inside the region scope. Heartbeat output may typically involve global, node, and element variables, and, on some occasions, face variables.

More than one heartbeat file can be specified for an analysis. For each heartbeat file, there will be one `HEARTBEAT OUTPUT` command block. The command block for a heartbeat file description begins with

```
BEGIN HEARTBEAT OUTPUT <string>heartbeat_name
```

and is terminated with

```
END [HEARTBEAT OUTPUT <string>heartbeat_name]
```

where `heartbeat_name` is a user-selected name for the command block. Nested within the `HEARTBEAT OUTPUT` command block are a set of command lines, as shown in the block summary given above. The first command line listed (`STREAM NAME`) gives pertinent information about the heartbeat file. The command line

```
STREAM NAME = <string>heartbeat_file_name
```

gives the name of the heartbeat file with the string `heartbeat_file_name`. A file that already exists is overwritten. If the heartbeat file is to appear in the current directory and is named `job.h`, this command line would appear as

```
STREAM NAME = job.h
```

If the heartbeat file is to be created in some other directory, the command line would have to show the absolute path to that directory. If the directory does not exist yet, all necessary directories are automatically created.

In addition to specifying a specific file name, there are several predefined streams that can be specified. The predefined streams are:

- `cout` or `stdout` specifies standard output;
- `cerr`, `stderr`, `clog`, or `log` specifies standard error;
- `output` or `outputP0` specifies Sierra's standard output which is redirected to the file specified by the `-o` option on the command line.

Two meta-characters can appear in the name of the heartbeat file. If the `%P` character is found in the name, it will be replaced with the number processors being used for the run. For example, if, in a given analysis on 1024 processors, the name `heartbeat-%P/job.h` is used, then the name would be expanded to `heartbeat-1024/job.h`. The other recognized meta-character is `%B` replaced with the base name of the input file containing the input commands. For example, if the commands are in the file `my_analysis_run.i` and the heartbeat stream name is specified as `%B.h`, then the heartbeat data would be written to the file `my_analysis_run.h`. If the stream name command line is omitted, the default file name will be `%B.hrt` which is the base name of the input file with the extension `.hrt`.

The other command lines that appear in the `HEARTBEAT OUTPUT` command block determine the type, frequency, and format of output information. Descriptions of these command lines follow in Section 9.5.1 through Section 9.5.8. The command lines for controlling the frequency of heartbeat output (in Section 9.3.1.9) are the same as those for controlling the frequency of results and history output. The `APPEND` command can be used to cause heartbeat information output by the current run to append to an existing file name rather than overwrite the existing file.

9.5.1 Output Variables

The `GLOBAL`, `NODE` (or `NODAL`), `EDGE`, `FACE`, or `ELEMENT` command line is used to select variables for output in the heartbeat file. One of several types of variables can be selected for output. The form of the command line varies depending on the variable type selected for output.

9.5.1.1 Global Output Variables

```
GLOBAL <string>variable_name
      [AS <string>heartbeat_variable_name]
```

This form of the command allows the user to select any global variable for output in the heartbeat file. The variable is selected with the string `variable_name`. The string `variable_name` is the name of the global variable and can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4). The `variable_name` can also specify `time`, `timestep`, or `step` to output the current simulation time, time step, or execution step, respectively.

A name, `heartbeat_variable_name`, for the selected entity can also be optionally specified following the `AS` keyword. For example, the CPU time (`cpu_time`) could be output as `cpu` in the

heartbeat file using the command:

```
GLOBAL cpu_time AS cpu
```

9.5.1.2 Mesh Entity Output Variables

```
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name
  AT NODE|NODAL|EDGE|FACE|ELEMENT <integer>entity_id
  AS <string>heartbeat_variable_name
```

This form of the command allows the user to select any nodal, edge, face, or element variable for a specific mesh entity for output in the heartbeat file. For example, this command line will pick the displacement at a specific node and output the displacement to the heartbeat file using a chosen alias.

For this form of the command, the mesh entity type is set to NODE (or NODAL), EDGE, FACE, or ELEMENT depending on the variable (set by `variable_name`) to be output. If the mesh entity type is set to NODE (or NODAL), EDGE, or FACE, the string `variable_name` can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4). If the mesh entity type is set to ELEMENT, the string `variable_name` can be a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4).

Selection of a specific mesh entity follows the AT keyword. A mesh entity type (NODE [or NODAL], EDGE, FACE, or ELEMENT) is specified, followed by the specific integer identifier, `entity_id`, for the mesh entity. A name, `heartbeat_variable_name`, for the selected entity can be specified following the AS keyword. For example, the accelerations at node 88 can be output using the command line:

```
NODE ACCELERATION AT NODE 88 AS accel_88
```

where `accel_88` is the name used for this heartbeat variable in the heartbeat file.

For face and edge variables, an additional method of specifying the specific face or edge entity is provided. This method selects the face or edge by specifying the element that the face or edge is a member of and the local face or edge number on that element. To use this method, the element id and the face ordinal are specified as `id.ordinal`. The numbering of an elements faces and edges is specified in the Exodus II users manual [1]. Note that the selected face or edge must exist in the mesh database as a face or edge on a sideset, or the face or edge must be created internally by the application.

As an example, the command line to obtain the pressure for the third face of element 42 for the heartbeat file would be:

```
FACE PRESSURE AT FACE 42.3 AS PRES_42.3
```

Either the keyword NODE or NODAL can be used for nodal quantities.

As an example, the command line to obtain the von Mises stress for element 1024 for the heartbeat file would be:

```
ELEMENT VON_MISES AT ELEMENT 1024 AS vm_1024
```

where `vm_1024` is the name used for this heartbeat variable in the heartbeat file.

9.5.1.3 Nearest Point Output Variables

```
NODE|NODAL|EDGE|FACE|ELEMENT <string>variable_name
NEAREST LOCATION <real>global_x,
    real<global_y>, real<global_z>
AS <string>heartbeat_variable_name
```

This form of the command allows the user to select any nodal, edge, face, or element variable for output in the heartbeat file using a nearest point criterion. The command line described in this subsection is an alternative to the command line described in the preceding section, Section 9.5.1.2, for obtaining heartbeat output. The command line in this section or the command line in Section 9.5.1.2 produces heartbeat files with variable information. The difference in these two command lines (Section 9.5.1.3 and Section 9.5.1.2) is simply in how the variable information is selected.

For the above form of the command, the mesh entity type is set to `NODE` (or `NODAL`), `EDGE`, `FACE`, or `ELEMENT` depending on the variable (set by `variable_name`) to be output. If the mesh entity type is set to `NODE` (or `NODAL`), `EDGE`, or `FACE`, the string `variable_name` can be either a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4). If the mesh entity type is set to `ELEMENT`, the string `variable_name` can be a variable listed in Section 9.10 or a user-defined variable (see Section 9.3.2 and Section 11.2.4).

The specific mesh entity used for output is determined by global coordinates specified by the `NEAREST LOCATION` keyword and its associated input parameters—`global_x`, `global_y`, `global_z`. The specific mesh entity chosen for output is as follows:

- If the mesh entity has been set to `NODE` (or `NODAL`), the node in the mesh selected for output is the node whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.
- If the mesh entity has been set to `EDGE`, the edge in the mesh selected for output is the edge with a center point (the average location of the two end points of the edge) whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.
- If the mesh entity has been set to `FACE`, the face in the mesh selected for output is the face with a centroid whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.
- If the mesh entity has been set to `ELEMENT`, the element in the mesh selected for output is the element with a centroid whose initial position is nearest the input global X, Y, and Z coordinates specified with the parameters `global_x`, `global_y`, and `global_z`.

In all the above cases, the original model coordinates are used when selecting the nearest entity, not the current coordinates.

A name, `heartbeat_variable_name`, for the selected entity may be specified following the `AS` keyword. As an example, accelerations at a node closest to the point with global coordinates (1012.0, 54.86, 103.3141) may be output with the command line:

```
NODE ACCELERATION
NEAREST LOCATION 1012.0, 54.86, 103.3141 AS accel_near
```

where `accel_near` is the name used for this heartbeat variable in the heartbeat file.

Either the keyword `NODE` or `NODAL` can be used for nodal quantities.

9.5.2 Outputting Heartbeat Data on a Node Set

It is commonly desired to output heartbeat data on a single-node node set. If a mesh file is slightly modified, the node and element numbers will completely change. The node associated with a node set, however, remains the same, i.e., the node in the node set retains the same initial geometric location with the same connectivity to other elements even when its node number changes. Therefore, we might want to specify the heartbeat output for a node set with a single node rather than with the global identifier for a node. This can easily be accomplished, as follows:

```
begin user output
  node set = nodelist_1
  compute global disp_ns_1 as average of nodal displacement
end

begin heartbeat output
  global disp_ns_1
end
```

If `nodelist_1` contains only a single node, the heartbeat output variable `disp_ns_1` will contain the displacement for the single node in the node set. If `nodelist_1` contains multiple nodes, the average displacement of the nodes will be output.

9.5.3 Specifying Time Steps for Output

All of the same output step control commands available for results output are also available for heartbeat output. See Section [9.3.1.9](#).

9.5.4 Synchronize Output

```
SYNCHRONIZE OUTPUT
```

The `SYNCHRONIZE OUTPUT` command can be used in the same manner it is used in the `exodus` output command block. See Section [9.3.1.10](#) for more details about this command.

9.5.5 Use Output Scheduler

```
USE OUTPUT SCHEDULER <string>scheduler_name
```

The `USE OUTPUT SCHEDULER` command can be used in the same manner it is used in the `exodus` output command block. See Section [9.3.1.11](#) for more details about this command.

9.5.6 Write Heartbeat On Signal

```
OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|
SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|
SIGKILL|SIGILL|SIGSEGV
```

The `OUTPUT ON SIGNAL` command line is used to initiate the writing of a heartbeat file when the system encounters the specified signal. The signal can either occur as the result of a system error, or the user can explicitly send the specified signal to the application (See the system documentation man pages for “signal” or “kill” for more information). Only one signal type in the list of signal types should be entered for this command line. These signals cause the code to terminate before the code can add any current heartbeat output (heartbeat output past the last heartbeat output time step) to the heartbeat file. If the code encounters the specified error type during execution, a heartbeat file will be written before execution is terminated. This command line can also be used to force the writing of a heartbeat file at some point during execution of the code. Suppose the command line

```
OUTPUT ON SIGNAL = SIGUSR2
```

is included in the input file. While the code is running, a user can execute (from the keyboard) the system command line

```
kill -s SIGUSR2 pid
```

to force the writing of a results file. In the above system command line, *pid* is the process identifier, which is an integer. The `OUTPUT ON SIGNAL` command line is primarily a debugging tool for code developers.

9.5.7 Heartbeat Output Formatting Commands

There are several command lines for the heartbeat section that modify the formatting of the heartbeat text output. The default output for the heartbeat data consists of a line beginning with a timestamp showing the current wall-clock time followed by multiple columns of data, for example:

```
Begin HeartBeat Region_1_Heartbeat
  Stream Name = output
  At Step 0, Increment = 10

  precision is 5

  global step
  global timestep as dt
  global time
  global total_energy as te
End
```

```
+ [12:18:51] step=240, dt=3.13933e-04, time=7.56578e-02, te=4.02795e-06
+ [12:18:51] step=250, dt=3.13933e-04, time=7.87971e-02, te=1.32125e-06
+ [12:18:51] step=260, dt=3.13933e-04, time=8.19365e-02, te=6.88142e-07
```



```

+[12:18:51] step=270, dt=3.13933e-04, time=8.50758e-02, te=3.93574e-06
+[12:18:52] step=280, dt=3.13933e-04, time=8.82151e-02, te=7.46609e-06
+[12:18:52] step=290, dt=3.13933e-04, time=9.13545e-02, te=1.03856e-05
+[12:18:52] step=300, dt=3.13933e-04, time=9.44938e-02, te=1.36822e-05
+[12:18:52] step=310, dt=3.13933e-04, time=9.76331e-02, te=1.64630e-05

```

The above example begins each line with a timestamp followed by five labeled data columns. The precision of the real data is 5. This example does not have a legend. This format can be modified with these commands.

9.5.7.1 CTH SpyHis output format

```
FORMAT = SPYHIS|ORIGINAL|CSV(CSV)
```

The `FORMAT` described the output format for the heartbeat file

- `FORMAT=SPYHIS` specifies a particular spaced format that can be processed with the CTH SpyHis application as post-processor for time-history data.
- `FORMAT=ORIGINAL` is a legacy format option. This format is similar to the default CSV option, but does not have a comment after the time stamp column.
- `FORMAT=CSV` specifies a comma separated value format. This is the default format.

9.5.7.2 Specify floating point precision

```
PRECISION = <integer>precision
```

By default, the real data is written with a precision of 5 which gives 5 digits following the decimal point. This can be altered with the `PRECISION` command. If the command line `PRECISION = 2` is specified, then the above data would look like:

```

Begin HeartBeat Region_1_Heartbeat
...
precision = 2
...
End

+[12:18:51] step=240, dt=3.14e-04, time=7.57e-02, te=4.03e-06
+[12:18:51] step=250, dt=3.14e-04, time=7.88e-02, te=1.32e-06
+[12:18:51] step=260, dt=3.14e-04, time=8.19e-02, te=6.88e-07

```

The precision applies to all real data; it is not possible to specify a different precision for each variable.

9.5.7.3 Specify Labeling of Heartbeat Data

```
LABELS = <string>OFF|ON
```

The above example shows the default output which consists of a label and the data separated by '='. The existence of the labels is controlled with the `LABELS` command. If `LABELS = OFF` is specified, then the above data would look like:

```
Begin HeartBeat Region_1_Heartbeat
...
labels = off
precision = 2
...
End

+[12:17:37] 240, 3.14e-04, 7.57e-02, 4.03e-06
+[12:17:37] 250, 3.14e-04, 7.88e-02, 1.32e-06
+[12:17:38] 260, 3.14e-04, 8.19e-02, 6.88e-07
```

9.5.7.4 Specify Existence of Legend for Heartbeat Data

```
LEGEND = <string>OFF|ON
```

Outputting data without labels can make it easier to work with the data in a spreadsheet program or other data manipulation program, but with no labels, it is difficult to determine what the data represents. The `LEGEND` output will print a line at the beginning of the heartbeat output identifying the data in each column. For example:

```
Begin HeartBeat Region_1_Heartbeat
...
legend = on
labels = off
precision = 2
...
End

+[12:17:37] Legend: step, dt, time, te
+[12:17:37] 240, 3.14e-04, 7.57e-02, 4.03e-06
+[12:17:37] 250, 3.14e-04, 7.88e-02, 1.32e-06
+[12:17:38] 260, 3.14e-04, 8.19e-02, 6.88e-07
```

9.5.7.5 Specify format of timestamp

```
TIMESTAMP FORMAT <string>"timestamp_format"
```

Each line of the heartbeat output is preceded by a timestamp which shows the wall-clock time at the time that the line was output. This can be useful to verify that the code is still running and producing output and to determine how fast the code is running. The default timestamp is in the format `[12:34:56]` specified by the format `[%H:%M:%S]`. If a different format is desired, it can be specified with the `TIMESTAMP FORMAT` command line. The format must be surrounded by double or single quotes, and the format is defined to be the string between the first single or double quote and the last matching quote type. If a modified format is desired, the documentation for the

UNIX `strftime` provides some details on time format specification. The example below shows a timestamp format delimited by `\ttlbrace` and `\ttrbrace`. The timestamp consists of an ISO-8601 date format followed by the current time.

```
...
    timestamp format "%F %H:%M:%S"
...
+{2008-03-17 09:26:17} 2212, 1.34244e-06, 2.96948e-03, 2.96948e-03
+{2008-03-17 09:26:17} 2213, 1.34244e-06, 2.97082e-03, 2.97082e-03
+{2008-03-17 09:26:17} 2214, 1.34244e-06, 2.97216e-03, 2.97216e-03
+{2008-03-17 09:26:17} 2215, 1.34244e-06, 2.97350e-03, 2.97350e-03
+{2008-03-17 09:26:17} 2216, 1.34244e-06, 2.97485e-03, 2.97485e-03
```

9.5.8 Monitor Output Events

```
MONITOR = <string>RESULTS|RESTART|HISTORY
```

It is sometimes a benefit to know when the code has written a new set of data to one of the other output files (restart output, history output, or results output). The heartbeat output will report this data if the `MONITOR` command line is specified. Each time output is performed to any of the monitored output types, a line will be written to the heartbeat file specifying the timestamp, the simulation time and step, and the label name of the output type. For example:

```
begin results output my_results
  at step 0, increment = 10
...
end results output results

begin heartbeat data hb
  stream name = stdout
  monitor = results
  labels = off
  legend = on
  timestamp format "%F %H:%M:%S "
  at step 0, increment = 2
  global step
  global timestep as dt
  global time
  element spring_engineering_strain at \#
    element 1 as sp1
end
```

Will give the output:

```
....
+2008-03-17 10:03:22 718, 1.34244e-06, 9.63871e-04, 9.63871e-04
```

```
-2008-03-17 10:03:22 Results data written at time = 0.00096656,  
step = 720. my_results  
+2008-03-17 10:03:22 720, 1.34244e-06, 9.66556e-04, 9.66556e-04  
+2008-03-17 10:03:22 722, 1.34244e-06, 9.69241e-04, 9.69241e-04  
+2008-03-17 10:03:22 724, 1.34244e-06, 9.71926e-04, 9.71926e-04  
+2008-03-17 10:03:22 726, 1.34244e-06, 9.74611e-04, 9.74611e-04  
+2008-03-17 10:03:22 728, 1.34244e-06, 9.77296e-04, 9.77296e-04  
-2008-03-17 10:03:22 Results data written at time = 0.00097998,  
step = 730. my_results  
+2008-03-17 10:03:22 730, 1.34244e-06, 9.79981e-04, 9.79981e-04  
.....
```

9.6 Restart Data

```
BEGIN RESTART DATA <string>restart_name
#
# Database Commands
DATABASE NAME = <string>restart_file
INPUT DATABASE NAME = <string>restart_input_file
OUTPUT DATABASE NAME = <string>restart_output_file
DATABASE TYPE = <string>database_type(exodusII)
OVERWRITE = <string>OFF|ON|TRUE|FALSE|YES|NO
            (ON|TRUE|YES)
PROPERTY = <string>propName = <string>propValue

RESTART TIME = <real>time
RESTART = AUTOMATIC

START TIME = <real>restart_start_time
TIMESTEP ADJUSTMENT INTERVAL = <integer>steps
AT TIME <real>time_begin INCREMENT =
    <real>time_increment_dt
ADDITIONAL TIMES = <real>output_time1
    <real>output_time2 ...
AT STEP <integer>step_begin INCREMENT =
    <integer>step_increment
ADDITIONAL STEPS = <integer>output_step1
    <integer>output_step2 ...
AT WALL TIME <real>time_begin[s|m|h|d] INCREMENT =
    <real>time_increment_dt[s|m|h|d]
TERMINATION TIME = <real>termination_time_value
OVERLAY COUNT = <integer>overlay_count
CYCLE COUNT = <integer>cycle_count
FILE CYCLE COUNT = <integer>file_cycle_count
SYNCHRONIZE_OUTPUT
USE OUTPUT SCHEDULER <string>scheduler_name
COMPONENT SEPARATOR CHARACTER = <string>character|NONE
OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|
    SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|
    SIGKILL|SIGILL|SIGSEGV
OPTIONAL
DECOMPOSITION METHOD = RCB|RIB|LINEAR|HSFC|BLOCK|CYCLIC|
    RANDOM|KWAY|GEOM_KWAY|KWAY_GEOM|METIS_SFC|EXTERNAL

END [RESTART DATA <string>restart_name]
```

🔗 **Explicit**

The restart capability in Sierra/SM allows a simulation to resume after the analysis has terminated. Restart can be used to break a long analysis into several smaller runs so the user can examine intermediate results before proceeding. In case of abnormal termination, such as a machine outage,

restart may be used to avoid rerunning the entire simulation. If a restart file has been written at various intervals throughout the analysis up to the point where the abnormal termination has occurred, the analysis can be restarted at a time prior to the abnormal termination, and the simulation can continue.

In general, the restart capability assumes the simulation input parameters are the same before and after the restart. Therefore, it is highly recommended that parameters such as boundary conditions, element types, and material models remain the same for at least the first step of each ‘restart run’.

It is also highly discouraged to restart from an implicit simulation directly into an explicit simulation or vice versa. If an implicit/explicit restart is desired, refer to Section 9.6.8 below.

Many input parameters may be modified in a restart, such as hourglass control or solver parameters. If an abnormal termination is encountered due to, for example, element inversion or an implicit solver convergence problem, it is recommended to modify these parameters to allow the simulation to continue past the abnormal termination time previously encountered.

The amount of data written at a restart time is large, since it contains a complete description of the state for the problem at that time. The restart data includes not only information such as displacement, velocity, and acceleration, but also information such as element stresses and all state variables for the material model associated with each element. Careful consideration should be used to choose the increment for writing restart files (so that storage space is not completely filled) and to determine if it is necessary to overwrite previous restart files.

The time at which to restart or automatic restart can be specified within the Sierra scope (see Section 2.1.3.1 for more details), but the same commands are available within the `RESTART DATA` command block as well.

In order to perform a basic restart, it is required to specify (i) the restart database name to read or write and (ii) either automatic restart or the time for restart to take place, for example:

```
BEGIN RESTART DATA RESTART_DATA
  DATABASE NAME = g.rsout
  Restart = auto
END RESTART DATA RESTART_DATA
```

Note that if this command is used, restart data will only be written at the last step of the analysis or if Sierra/SM detects an internal error such as element inversion. To specify more frequent times to write restart data, the `AT TIME ...` or `AT STEP ...` command should be used.

```
BEGIN RESTART DATA RESTART_DATA
  DATABASE NAME = g.rsout
  Restart = auto
  At Time 1e-4 Increment = 1e-5
END RESTART DATA RESTART_DATA
```

For more complex uses of restart, please read below for further details.

9.6.1 Restart Options

The `RESTART DATA` command block begins with the input line:

```
BEGIN RESTART DATA <string>restart_name}
```

and is terminated with:

```
END [RESTART DATA <string>restart_name]
```

where `restart_name` is a user-selected name for the `RESTART DATA` command block.

Nested within the `RESTART DATA` command block are a set of command lines, as shown in the block summary given above.

```
DATABASE NAME = <string>restart_file
INPUT DATABASE NAME = <string>restart_input_file
OUTPUT DATABASE NAME = <string>restart_output_file
DATABASE TYPE = <string>database_type(exodusII)
OPTIONAL
```

The database name commands are used to specify the names of the files to read restart data from or write restart data to. If the `OUTPUT DATABASE NAME` command is used, restart data will be written to the filename specified. If the `INPUT DATABASE NAME` command is used, restart data will be read from the file with the specified filename. If the file does not exist at the specified path, the simulation will start from the beginning. If the `DATABASE NAME` command is used, the filename specified will be used for both writing out restart data and reading in restart data. Either the `DATABASE NAME` command should be used alone or the `INPUT DATABASE NAME` command line/`OUTPUT DATABASE NAME` command line should be used, but not both. The `DATABASE NAME` command is more automated than specifying individual input and output database names and is the preferred method. Section 9.6.1.2 through Section 9.6.1.5 shows how this command line is used in specific instances.

See Section 9.2 for more details on the general database commands such as `DATABASE NAME`, `DATABASE TYPE`, `OVERWRITE`, etc.

In coupled physics analyses with multiple regions, only a subset of the regions may have a restart database associated with them. The `OPTIONAL` command (Section 9.6.1) is used to tell the application that it is acceptable to restart the analysis even though a region does not have an associated restart database. This is only allowed in analyses containing multiple regions; if there is only a single region, it must have a restart database to restart.

9.6.1.1 What Times to Write

Several options exist to write restart at specific points in the analysis.

- The `AT TIME` and `ADDITIONAL TIMES` commands define restart in terms of analysis time
- The `AT STEP` and `ADDITIONAL STEPS` commands define restart in terms of analysis step number

- The `AT WALL TIME` command defines restart in terms of wall clock runtime. An example of the expected format is

```
AT WALL TIME 3600s INCREMENT = 60m
```

This example would write the first restart after 3600 seconds of runtime; then additional restart data will be written every 60 minutes thereafter. The units of time must be specified by `s` (seconds), `m` (minutes), `h` (hours), or `d` (days).

Note that restart can only be written at the end of a completed step. If a restart time t_s which does not coincide with a simulation time step t_i is specified, restart data will be written at the next analysis step $t_{n+1} \geq t_s$. For example, if an analysis has steps at times 0.1 and 0.2, asking for restart at time 0.125 will write the restart data at time 0.2 (the end of the next completed step). Additionally, restart will only be written at the end of a completed implicit loadstep.

9.6.1.2 Automatic Read and Write of Restart Files

Restart files can be tracked in an automated fashion by using the `RESTART = AUTO` command. Automatic restart can best be explained by an example. For the below example, assume the problem is run on two processors and all files are in the current working directory.

The automated restart option will not only manage the restart files to prevent overwriting, it will also manage the results files and history files to prevent overwriting. In the example given, assume the results file is defined with the command line

```
DATABASE NAME = rslt.e
```

where this command will generate the file `rslt.e`. Also assume history output is defined with the command line

```
DATABASE NAME = hist.h
```

to generate the file `hist.h`.

For all runs in this example, the command line

```
RESTART = AUTOMATIC
```

should be in either the `SIERRA` scope or the `RESTART DATA` command block. The `RESTART TIME` command line should not be used with `RESTART = AUTOMATIC`. For additional documentation of the `RESTART TIME` command, refer to Section [2.1.3.1](#).

Finally, assume the `RESTART DATA` command block is defined as follows:

```
BEGIN RESTART DATA RESTART_DATA
  DATABASE NAME = g.rsout
  AT TIME 0.0 INCREMENT = 0.25E-3
  RESTART = AUTO      #Optionally placed here or SIERRA scope
END RESTART DATA RESTART_DATA
```


In this block, the `DATABASE NAME` command line specifies a root file name for the restart file. The `AT TIME` command line gives the time and the increment for which restart information will be written.

Assume the first job is launched; the first run will generate the following output files:

```
# restart files
g.rsout.2.0
g.rsout.2.1
# results files
rslt.e.2.0
rslt.e.2.1
# history files
hist.h.2.0
hist.h.2.1
```

For the above files, the `.2.0` and `.2.1` extensions designate how the mesh was decomposed across the two processors.

Now, it is desired to start a second simulation using the restart data from the first, at the termination time of the previous run. All of the restart information in the input file will remain the same. In the `TIME CONTROL` command block, the only line that needs to be changed is the `TERMINATION TIME` command line. Even if the `START TIME` remains at `0.0`, this time will be superseded by the last output time in the restart file.

The second job may then be launched; when it is completed, the following output files will be in the current working directory:

```
# restart files
g.rsout.2.0
g.rsout.2.1
g.rsout-s0002.2.0
g.rsout-s0002.2.1
# results files
rslt.e.2.0
rslt.e.2.1
rslt.e-s0002.2.0
rslt.e-s0002.2.1
# history files
hist.h.2.0
hist.h.2.1
hist.h-s0002.2.0
hist.h-s0002.2.1
```

All of the files that were previously written remain untouched; new files with the extension `-s0002` contain the output data associated with the second run.

The process outlined above can be continued as long as necessary. As long as the `RESTART = AUTO` command remains, new output files with the `-s*` extension will continue to be generated and the previous files will *not* be overwritten.

9.6.1.3 User-Controlled Read and Write of Restart Files

Restart times and input/output database names may be specified using a combination of the `RESTART TIME` command line in the `SIERRA` scope and the `INPUT DATABASE NAME` and `OUTPUT DATABASE NAME` command lines in the `RESTART DATA` command block, allowing the user fine-grained control of restart data.

Note, however, that restart information is always automatically written upon element inversion.

User-controlled restart can best be explained by an example. We will use a two-processor example and assume all files will be in our current directory. In this example, the creation of new restart files is managed by the user so as not to overwrite existing restart files, analogously to the automated option. Unlike the automated option for restart, this controlled use of restart requires the user to manage restart file names manually so as to prevent overwriting previously generated files. The user must also manage the creation of new results and history output files so as not to overwrite existing files by creating new results/history files for each run in the sequence of restart runs requires changing the `DATABASE NAME` command line in the `RESULTS OUTPUT` and `HISTORY OUTPUT` command blocks. Explicit examples of `DATABASE NAME` command line usage in the `RESULTS OUTPUT` and `HISTORY OUTPUT` command blocks are omitted, since it closely parallels the pattern managing restart file names.

For the first run in the restart sequence, only a `RESTART DATA` command block is present in the region; there is no restart-related command line in the `SIERRA` scope of the input file. However, a

```
START TIME = 0.0
```

command line in a `TIME STEPPING` command block (within the `TIME CONTROL` command block) and a

```
TERMINATION TIME = 2.5E-3
```

command line within the `TIME CONTROL` command block are present to set the limits for the begin and end times. The `RESTART DATA` command block in the example input file is as follows:

```
BEGIN RESTART DATA RESTART_DATA
  OUTPUT DATABASE NAME = RS1.rsout
  AT TIME 0.0 INCREMENT = 0.5E-3
END RESTART DATA RESTART_DATA
```

In the first run, the restart option will generate the restart files:

```
RS1.rsout.2.0
RS1.rsout.2.1
```

The `.2.0` and `.2.1` file name extensions correspond to each processor's mesh file in a two-processor run. If our mesh file is `mesh.g`, then our mesh files on the individual processors will be `mesh.g.2.0` and `mesh.g.2.1`. All restart information in the above files appears at time intervals of 0.5×10^{-3} , and the last restart information is written at time 2.5×10^{-3} .

For the second run in the sequence of restart runs, it is desired to start at the previous termination time, 2.5×10^{-3} , and terminate at time 5.0×10^{-3} . To do this, the command line

```
RESTART TIME = 2.5E-3
```

must be added to the SIERRA scope, and the termination time must be set using the command line

```
TERMINATION TIME = 5.0E-3
```

within the `TIME CONTROL` command block.

The actual start time for the second run in our analysis is now set by the restart time set on the `RESTART TIME` command line, 2.5×10^{-3} . The command line `START TIME = 0.0` in the `TIME STEPPING` command block is now superseded as the actual starting time for the second run by the restart commands. Any `START TIME` command line in a `TIME STEPPING` command block is still valid for defining time stepping blocks (used to set activation periods), but the restart process sets the actual start time for our analysis. This pattern of control for setting the actual start time holds for any run in the sequence of restart runs.

The `RESTART DATA` command block must also be modified to read:

```
BEGIN RESTART DATA RESTART_DATA
  INPUT DATABASE NAME = RS1.rsout
  OUTPUT DATABASE NAME = RS2.rsout
  AT TIME 0.0 INCREMENT = 0.5E-3
END RESTART DATA RESTART_DATA
```

For the second run, the files

```
RS1.rsout.2.0
RS1.rsout.2.1
```

are read, and the files

```
RS2.rsout.2.0
RS2.rsout.2.1
```

are written. All restart information in `RS2.rsout.2.0` and `RS2.rsout.2.1` appears at time intervals of 0.5×10^{-3} , restart information is written from times 2.5×10^{-3} to 5.0×10^{-3} . The restart files from the first run of the restart sequence have been preserved, since distinct input and output database names, `RS2.rsout` and `RS1.rsout`, respectively, have been specified.

Now, a third run is conducted in the restart sequence, starting at time 4.5×10^{-3} and terminating at time 8.5×10^{-3} . The second run terminated at time 5.0×10^{-3} ; thus, a restart time must be manually specified:

```
RESTART TIME = 4.5E-3
```

and the `TERMINATION TIME` command line within the `TIME CONTROL` command block must be specified:

```
TERMINATION TIME = 8.5E-3
```

The `RESTART DATA` command block should also read:

```
BEGIN RESTART DATA RESTART_DATA
  INPUT DATABASE NAME = RS2.rsout
  OUTPUT DATABASE NAME = RS3.rsout
  AT TIME 0.0, INCREMENT = 0.5E-3
END RESTART DATA RESTART_DATA
```

The third run will read restart files:

```
RS2.rsout.2.0
RS2.rsout.2.1
```

and write restart data to files:

```
RS3.rsout.2.0
RS3.rsout.2.1
```

All restart information in `RS3.rsout.2.0` and `RS3.rsout.2.1` appears at time intervals of 0.5×10^{-3} , and restart information is written from times 4.5×10^{-3} to 8.5×10^{-3} . The restart files from the first and second runs of the restart sequence have been preserved, since distinct input and output database names have been specified.

9.6.1.4 Overwriting Restart Files

By default, every time an analysis is restarted, that output file the restart data is written to will be overwritten. As previously stated, it is best to have a separate restart file (or files for parallel runs) associated with each run in a sequence of restart runs if storage space permits. To not overwrite a restart file, the `OVERWRITE = OFF` command should be used. If this command is used and a restart file already exists with the name given in the `DATABASE NAME` command line, Sierra/SM will throw an error on start-up. To fix this error, the name in the `DATABASE NAME` command should be changed.

The below example demonstrates how the restart data will be written:

For the first run, suppose the termination time is set as

```
TERMINATION TIME = 1.0E-3
```

and the `RESTART DATA` command block is as follows:

```
BEGIN RESTART DATA
  DATABASE NAME = RS.out
  AT TIME 0.0 INTERVAL = 0.25E-3
END RESTART DATA
```

The above command block will output restart data to the file `RS.out` at intervals of 0.25×10^{-3} from time 0.0 through the termination time.

For the second run, suppose the termination time is set as:

```
TERMINATION TIME = 2.0E-3
```

and the restart data block is:

```
BEGIN RESTART DATA
  DATABASE NAME = RS.out
  AT TIME 0.0 INTERVAL = 0.25E-3
  RESTART TIME = 1.0E-3
END RESTART DATA
```

For this second run, restart information is read from the `RS.out` file(s) at time 1.0×10^{-3} . Then, these files are overwritten with new restart data generated starting at time 1.0×10^{-3} at intervals of 0.25×10^{-3} .

This pattern will continue for all analyses run in the set of restart runs.

9.6.1.5 Recovering from a Corrupted Restart

When performing an analysis during a sequence of restart runs, it is sometimes possible that a restart file becomes corrupted (due to a system crash for example). If the `RESTART=AUTO` option is exercised, restart will detect the corrupted entry and find the closest previous valid entry which can be used for restart.

A manual recovery can also be performed using the `RESTART TIME` command. Using this method, the user must select a restart time before the file corruption occurred. The `INPUT DATABASE NAME` and `OUTPUT DATABASE NAME` command lines should be used to avoid overwriting previous restart files. Additionally, the filenames should also be changed in the results and history command blocks to avoid overwriting those files.

9.6.2 Component Separator Character

```
COMPONENT SEPARATOR CHARACTER = <string>character|NONE
```

The `COMPONENT SEPARATOR CHARACTER` command can be used in the same manner it is used in the exodus output command block. See Section 9.3.1.7 for more details about this command.

9.6.3 Specifying Time Steps for Output

All of the same output step control commands available for results output are also available for restart output. See Section 9.3.1.9.

9.6.4 Reducing Restart File Size

As stated in the introduction, the amount of data written for restart output is significant; therefore, restart files can become very large. The commands in this section may be used to limit restart file size. These commands may be used individually or collectively.

9.6.4.1 Overlay Count

```
OVERLAY COUNT = <integer>overlay_count
```

The `OVERLAY COUNT` command specifies the number of restart output times that will be overlaid on top of the current step before advancing to the next step. For example, suppose the `overlay_count` parameter $n_o = 2$, and restart data is set to be written every $\Delta t_s = 0.1$ second. At time 0.1 seconds, restart step 1 will be written to the output restart database. At time 0.2 seconds, restart information will be written over the step 1 information, which originally contained restart information at 0.1 seconds; similarly, step 1 information will once again be overwritten at time 0.3 seconds. At time 0.4 seconds, step 1 will now be complete and data will be written to step 2. At time 0.5 seconds, restart information will be written over the step 2 information, which originally contained information at 0.4 seconds. This process then continues throughout the simulation.

9.6.4.2 Cycle Count

```
CYCLE COUNT = <integer>cycle_count  
FILE CYCLE COUNT = <integer>file_cycle_count
```

The `CYCLE COUNT` command line specifies the number of restart steps that will be written to the output restart database before previously written steps are overwritten. For example, suppose the `cycle_count` parameter $n_c = 5$, and restart data is set to be written every $\Delta t_s = 0.1$ second. The restart system writes information to the output restart database at times 0.1, 0.2, ..., 0.5 seconds. The information at step 1 (originally written at time 0.1) is overwritten with information at time 0.6. At time 0.7, step 2 is overwritten, etc. At time 0.8, the output restart database will contain information at times 0.6, 0.7, 0.8, 0.4, and 0.5 seconds—time will not necessarily be monotonically increasing on a restart database that uses `CYCLE COUNT`. If it is desired to only keep the last step available on the output restart database, `CYCLE_COUNT` should be set to 1.

9.6.4.3 Example

As stated previously, the `CYCLE COUNT` and `OVERLAY COUNT` command lines can be used at the same time. The following example is used to explain that scenario:

It is assumed the overlay count $n_o = 2$ and the cycle count $n_c = 5$. Information is written to the output restart database time step every $\Delta t_s = 0.1$ second. The times at which information is written to the output restart database are $t_n = n\Delta t_s$, $n = 1, 2, \dots, N$ seconds, where n is an output step and N is the number of steps. The overlay command will result in information at times $t_i = i(n_o + 1)\Delta t_s$, $i = 1, 2, \dots, n_c$ seconds written at steps i on the output restart database—in the example, times 0.3, 0.6, 0.9, 1.2, and 1.5, at steps 1, 2, 3, 4, and 5. For $t_i > n_c(n_o + 1)\Delta t_s$, or 1.5 seconds, the cycle command will take effect because $n_c = 5$ different time steps are currently written on the output restart database. The overlay procedure then resumes at time $t = (1 + n_c(n_o + 1))\Delta t_s$: information at subsequent times 1.6, 1.7, and 1.8 seconds overwrite the information at step 1, which previously held information at time 0.3 second. Information at times 1.9, 2.0, and 2.1 seconds overwrites the information at step 2, etc. For any output step n , its corresponding restart

output database step number n_s is given by

$$n_s = 1 + \frac{(n - 1) \bmod n_c(n_o + 1)}{n_o + 1},$$

where $i \bmod j$ represents the modulo operator, or the remainder of i/j . The quotient is evaluated using integer arithmetic (any fractional remainder is discarded). For example, if $n = 4$, $n_c = 5$, and $n_o = 2$, then $(4 - 1) \bmod 5(2 + 1) = 3$ and $n_s = 1 + 3/3 = 2$.

The `FILE CYCLE COUNT` command is related to `CYCLE COUNT`; using this option, each restart dump will be written to a separate file suffixed with A, B, etc., where the count specifies how many separate files are written before the cycle repeats. For example, if `FILE CYCLE COUNT = 3` would cycle through `file-A.rs`, `file-B.rs`, `file-C.rs`, `file-A.rs`, etc. The maximum value for the cycle count is 26.

9.6.5 Synchronize Output

```
SYNCHRONIZE OUTPUT
```

The `SYNCHRONIZE OUTPUT` command can be used in the same manner it is used in the `exodus` output command block. See Section 9.3.1.10 for more details about this command.

9.6.6 Use Output Scheduler

```
USE OUTPUT SCHEDULER <string>scheduler_name
```

The `USE OUTPUT SCHEDULER` command can be used in the same manner it is used in the `exodus` output command block. See Section 9.3.1.11 for more details about this command.

9.6.7 Write Restart If System Error Encountered

```
OUTPUT ON SIGNAL = <string>SIGALRM|SIGFPE|SIGHUP|SIGINT|
SIGPIPE|SIGQUIT|SIGTERM|SIGUSR1|SIGUSR2|SIGABRT|
SIGKILL|SIGILL|SIGSEGV
```

The `OUTPUT ON SIGNAL` command line is used to initiate the writing of a restart file when the computer system encounters an error. Only one error type in the list of error types should be entered for this command line. These system errors cause the code to terminate before the code can add any current restart output (restart output past the last restart output time step) to the restart file. If the code encounters the specified error type during execution, a restart file will be written before execution is terminated. This command line can also be used to force the writing of a restart file at some point during execution of the code. Suppose the command line

```
OUTPUT ON SIGNAL = SIGUSR2
```

is included in the input file. While the code is running, a user can execute (from the keyboard) the system command line

```
kill -s SIGUSR2 {pid}
```

to terminate execution and force the writing of a results file. In the above system command line, `pid` is the process identifier, an integer.

The most useful application of the command line is to send a signal via a system command line to write a restart file. The `OUTPUT ON SIGNAL` command line is primarily a debugging tool for code developers.

9.6.8 Restarting an Implicit/Explicit Analysis

Restarting from an implicit analysis directly into an explicit analysis (or vice versa) is not expected to work correctly; Sierra/SM uses different default settings for the two simulation types, and internal modifications are required when transitioning from a simulation which does not consider mass/inertial effects (implicit quasistatics) to one that does (explicit dynamics), or vice versa. To start an explicit simulation from a restart file generated by an implicit simulation, for example, the following procedure should be followed:

Suppose the input file used to generate the implicit restart file is called `implicit.i`, and the file that contains the input commands for the explicit simulation is called `explicit.i`. Before restarting into the explicit simulation, the contents of the `adagio` procedure in `implicit.i` must be copied and pasted into the procedure before the `presto` procedure in `explicit.i`—the procedures should have different names. If using the `RESTART=AUTO` option, the restart block in the implicit procedure may remain unchanged. If manually specifying the time at which to restart, the `RESTART TIME` should be set to the termination time of the implicit procedure (prior time, if desired). A procedural transfer block should then be added to the `adagio` procedure performing the explicit dynamics portion of the run. The simplest and most conventional procedural transfer block for this purpose is as follows:

```
BEGIN PROCEDURAL TRANSFER trans
  INCLUDE ALL BLOCKS
END
```

For more information on procedural transfers, see Section [12.2](#). This pattern should always be followed when restarting from different analysis types (e.g., Implicit Quasistatics to Implicit Dynamics, Explicit Dynamics to Implicit Quasistatics, Explicit Dynamics to Implicit Dynamics, etc.).

9.7 Output Scheduler

In an analysis with multiple regions, it can be difficult to synchronize output such as results files, history files, and restart files. To help synchronize output for analyses with multiple regions, an `OUTPUT SCHEDULER` command block may be defined at the SIERRA scope. This scheduler can then be referenced

- in the `RESULTS OUTPUT` command block to control the output of results information,
- in the `HISTORY OUTPUT` command block to control the output of history information,
- in the `HEARTBEAT OUTPUT` command block to control the output of heartbeat information, and/or
- in the `RESTART DATA` command block to control the writing of restart files.

The `OUTPUT SCHEDULER` command block is defined in the SIERRA scope and is referenced by a `USE OUTPUT SCHEDULER` command line inside the appropriate output block. Section 9.7.1 describes the `OUTPUT SCHEDULER` command block, and Section 9.7.2 illustrates how this block is referenced with the `USE OUTPUT SCHEDULER` command line.

9.7.1 Output Scheduler Command Block

```
BEGIN OUTPUT SCHEDULER <string>scheduler_name
  START TIME = <real>output_start_time
  TIMESTEP ADJUSTMENT INTERVAL = <integer>steps
  AT TIME <real>time_begin INCREMENT =
    <real>time_increment_dt
  ADDITIONAL TIMES = <real>output_time1
    <real>output_time2 ...
  AT STEP <integer>step_begin INCREMENT =
    <integer>step_increment
  ADDITIONAL STEPS = <integer>output_step1
    <integer>output_step2 ...
  TERMINATION TIME = <real>termination_time_value
END [OUTPUT SCHEDULER <string>scheduler_name]
```

↪ Explicit

An output scheduler is defined with a command block in the SIERRA scope. The `OUTPUT SCHEDULER` command block begins with the input line:

```
BEGIN OUTPUT SCHEDULER <string>scheduler_name
```

and is terminated with the line:

```
END OUTPUT SCHEDULER <string>scheduler_name
```

where `scheduler_name` is a user-defined name for the command block. All the normal scheduling command lines are valid in an `OUTPUT SCHEDULER` command block.

9.7.1.1 Specifying Time Steps for Output

All of the same output step control commands available for results output are also available in the output scheduler. See Section 9.3.1.9.

9.7.2 Example of Using the Output Scheduler

Once an output scheduler has been defined via the `OUTPUT SCHEDULER` command block, it can be used by inserting a `USE OUTPUT SCHEDULER` command line in any of the command blocks: `RESULTS OUTPUT`, `HISTORY OUTPUT`, and `RESTART DATA`. The next paragraph provides an example of using output schedulers.

In the SIERRA scope, we define two output schedulers, `Timer` and `Every_Step`:

```
BEGIN OUTPUT SCHEDULER Timer
  AT TIME 0.0 INCREMENT = 10.0e-6
  TIME STEP ADJUSTMENT INTERVAL = 4
END OUTPUT SCHEDULER Timer
#
BEGIN OUTPUT SCHEDULER Every_Step
  AT STEP 0 INCREMENT = 1
END OUTPUT SCHEDULER Every_Step
```

With the `USE OUTPUT SCHEDULER` command, we reference the scheduler named `Timer` for results output:

```
BEGIN RESULTS OUTPUT Out_Region_1
.
  USE OUTPUT SCHEDULER Timer
.
END RESULTS OUTPUT Out_Region_1
```

With the `USE OUTPUT SCHEDULER` command, we reference the scheduler named `Every_STEP` for history output:

```
BEGIN HISTORY OUTPUT Out_Region_2
.
  USE OUTPUT SCHEDULER Every_Step
.
END HISTORY OUTPUT Out_Region_2
```

9.8 Variable Interpolation

The variable interpolation command block is used to transfer data from one block or surface to another. Generally the transferred data is output data that will be used for a subsequent analysis. As an example the variable interpolation command block was created primarily to aid acoustic analysis where the derivative of void element volume is mapped from the structural Sierra/SM mesh to a background acoustic mesh for use in Salinas.

```
BEGIN VARIABLE INTERPOLATION [<string>var_interp_name]
  TRANSFER TYPE = INTERPOLATE FROM NEAREST FACE|
    SUM TO NEAREST ELEMENT
  SOURCE VARIABLE = ELEMENT|NODAL|
    SURFACE NORMAL NODAL <string>source_var_name
  SOURCE ELEMENT BLOCK = <string>source_block
  SOURCE SURFACE = <string>source_surface
  TARGET VARIABLE = ELEMENT|FACE|NODAL
    <string>target_var_name
  TARGET DERIVATIVE VARIABLE = ELEMENT|FACE|NODAL
    <string>target_deriv_var_name
  TARGET SURFACE = <string>target_surface
  SEARCH TOLERANCE = <real>search_tol
  PROXIMITY SEARCH TYPE = RAY SEARCH|SPHERE SEARCH
  RAY SEARCH DIRECTION = <real>vecx <real>vecy <real>vecz
  ONE WAY RAY SEARCH DIRECTION = <real>vecx <real>vecy <real>vecz
  RAY SEARCH DIRECTION = ORTHOGONAL TO LINE <real>p1x
    <real>p1y <real>p1z <real>p2x <real>p2y <real>p2z
  RAY SEARCH DIRECTION = TARGET SURFACE NORMAL

  ACTIVE PERIODS = <string list>period_names
  INACTIVE PERIODS = <string list>period_names
END [VARIABLE INTERPOLATION <string>var_interp_name]
```

The command TRANSFER TYPE can be set to either INTERPOLATE FROM NEAREST FACE or SUM TO NEAREST ELEMENT. The TRANSFER TYPE restricts the commands that can be used in conjunction with it.

When TRANSFER TYPE is set to INTERPOLATE FROM NEAREST FACE the available commands are restricted to:

```
BEGIN VARIABLE INTERPOLATION [<string>var_interp_name]
  TRANSFER TYPE = INTERPOLATE FROM NEAREST FACE
  SOURCE VARIABLE = NODAL|SURFACE NORMAL NODAL
    <string>source_var_name
  SOURCE SURFACE = <string>source_surface
  TARGET VARIABLE = NODAL <string>target_var_name
  TARGET DERIVATIVE VARIABLE = NODAL <string>target_deriv_var_name
  TARGET SURFACE = <string>target_surface
```

```

SEARCH TOLERANCE = <real>search_tol
PROXIMITY SEARCH TYPE = RAY SEARCH|SPHERE SEARCH
RAY SEARCH DIRECTION = <real>vecx <real>vecy <real>vecz
ONE WAY RAY SEARCH DIRECTION = <real>vecx <real>vecy <real>vecz
RAY SEARCH DIRECTION = ORTHOGONAL TO LINE <real>p1x
    <real>p1y <real>p1z <real>p2x <real>p2y <real>p2z
RAY SEARCH DIRECTION = TARGET SURFACE NORMAL
END [VARIABLE INTERPOLATION <string>var_interp_name]

```

For `TRANSFER TYPE = INTERPOLATE FROM NEAREST FACE`, a given point on the target surface is mapped to the closest point on the source surface. At that closest source point the source variable is interpolated. The interpolated value is then copied from the interpolated source point to the target point. Optionally if the `SURFACE NORMAL NODAL` command is being used and the variable being transferred is a vector then only the surface normal component of that variable will be transferred. This is useful for several applications involving transferring solid mesh quantities such as velocity to a reference mesh modeling a fluid (e.g., air). In such applications only the surface normal vibration is relevant to the air acoustic solve.

When `TRANSFER TYPE` is set to `SUM TO NEAREST ELEMENT` the available commands are restricted to:

```

BEGIN VARIABLE INTERPOLATION [<string>var_interp_name]
    TRANSFER TYPE = SUM TO NEAREST ELEMENT
    SOURCE VARIABLE = ELEMENT <string>source_var_name
    SOURCE ELEMENT BLOCK = <string>source_block
    TARGET VARIABLE = ELEMENT <string>target_var_name
    TARGET DERIVATIVE VARIABLE = ELEMENT
        <string>target_deriv_var_name
    TARGET SURFACE = <string>target_surface
    SEARCH TOLERANCE = <real>search_tol
    PROXIMITY SEARCH TYPE = RAY SEARCH|SPHERE SEARCH
    RAY SEARCH DIRECTION = <real>vecx <real>vecy <real>vecz
    ONE WAY RAY SEARCH DIRECTION = <real>vecx <real>vecy <real>vecz
    RAY SEARCH DIRECTION = ORTHOGONAL TO LINE <real>p1x
        <real>p1y <real>p1z <real>p2x <real>p2y <real>p2z
END [VARIABLE INTERPOLATION <string>var_interp_name]

```

For `TRANSFER TYPE = SUM TO NEAREST ELEMENT`, a given element in the source block computes its centroid. Then the closest face to that centroid on the target surface is found. The source element value is summed to the element attached to that target face.

The command `SOURCE VARIABLE` specifies the variable type, `ELEMENT`, `GLOBAL`, or `NODAL`, and the name of the source variable `source_var_name`.

The command `SOURCE SURFACE` specifies the name of the source side set `source_surface`.

The command `TARGET VARIABLE` specifies the variable type, `ELEMENT`, `FACE`, or `NODAL`, and the name of the target variable `target_var_name`. The target variable must exist. If necessary, a target variable can be created using the `BEGIN USER VARIABLE` command block (Section 11.2.4).

The command `TARGET DERIVATIVE VARIABLE` can be used to optionally compute the time derivative of the target variable. This command specifies the type of variable, `ELEMENT`, `FACE`, or `NODAL`, and the name of the variable to store the derivative, `target_deriv_var_name`. The target derivative variable must exist. If necessary, a target variable can be created using the `BEGIN USER VARIABLE` command block (Section 11.2.4).

The command `TARGET SURFACE` specifies the name of the target side set `target_surface`.

The command `SEARCH TOLERANCE` specifies a search distance for use in the proximity search.

The command `PROXIMITY SEARCH TYPE` can be set to either `RAY SEARCH` or `SPHERE SEARCH`. When `PROXIMITY SEARCH TYPE = SPHERE SEARCH` the `SEARCH TOLERANCE` is used as the radius for closest point searches. When `PROXIMITY SEARCH TYPE = RAY SEARCH` the `SEARCH TOLERANCE` and any `RAY SEARCH DIRECTION` are used together to form rays with lengths equal to the `SEARCH TOLERANCE`. If the `RAY SEARCH DIRECTION` command is used then the rays will extend in both the positive and negative directions. If the `ONE WAY RAY SEARCH DIRECTION` is used the rays will extend in only the positive direction. The one way rays are currently only available with the explicitly defined search direction vector. Faces penetrated by a ray are included in the set of faces for determining the closest point.

The commands `RAY SEARCH DIRECTION` and `ONE WAY RAY SEARCH DIRECTION` specify a search direction for the `PROXIMITY SEARCH = RAY SEARCH DIRECTION` case. A `RAY SEARCH DIRECTION` or `ONE WAY RAY SEARCH DIRECTION` can be specified directly with three values, `vecx`, `vecy`, and `vecz`. Alternatively, `RAY SEARCH DIRECTION` can be calculated through options `ORTHOGONAL TO LINE` or `TARGET SURFACE NORMAL`. For `ORTHOGONAL TO LINE` the search direction is the vector between a given point and the closest point on the infinite line specified with the two points `p1x`, `p1y`, `p1z` and `p2x`, `p2y`, `p2z`. For `TARGET SURFACE NORMAL` the search direction is the nodal normal vector calculated from the `TARGET SURFACE`.

9.8.1 Variable Interpolation Examples

The variable interpolation commands are complicated and best explained by the following example of a likely use case for this capability.

Example 1: Transferring volume infusion term to acoustic analysis. The example geometry is shown in Figure 9.3a run with the below syntax.

```
BEGIN VARIABLE INTERPOLATION
  TRANSFER TYPE = SUM TO NEAREST ELEMENT
  SOURCE VARIABLE = ELEMENT volume_first_derivative
  SOURCE ELEMENT BLOCK = block_1
  TARGET VARIABLE = ELEMENT volume_change_rate
  TARGET SURFACE = surface_2
  SEARCH TOLERANCE = 1.0e-3
END [VARIABLE INTERPOLATION <string>var_interp_name]
```

The volume derivatives are computed void elements (red) in source block one. At the centroid of each of these elements the closest facet on the target surface `surface_2` is found (green). The

volume derivatives are then summed to the elements attached to the target surface faces (following blue arrows). In this way the change of volume of structural void elements can be mapped to gas source term in a subsequent Salinas acoustic analysis. A similar boundary condition can be used to map structural energy source terms for a subsequent thermal analysis.

Example 2: Transferring surface motion for acoustic analysis. The example geometry is shown in Figure 9.3b run with the below syntax.

```
BEGIN VARIABLE INTERPOLATION [<string>var_interp_name]
  TRANSFER TYPE = INTERPOLATE FROM NEAREST FACE
  SOURCE VARIABLE = SURFACE NORMAL NODAL velocity
  SOURCE SURFACE = surface_1
  TARGET VARIABLE = NODAL surface_motion
  TARGET SURFACE = surface_2
  SEARCH TOLERANCE = 1.0e-3
  PROXIMITY SEARCH TYPE = RAY SEARCH|SPHERE SEARCH
END [VARIABLE INTERPOLATION <string>var_interp_name]
```

The source surface velocities (solid red arrows) are first restricted to the surface normal (dotted red arrows). These normal velocities define a velocity distribution on the source surface (thin red line). The velocity distribution at each node of the target surface (green arrows) found by spatial interpolation of the source surface normal velocity. In this way the solid normal surface motion can be mapped as a boundary condition on a subsequent Salinas acoustic analysis at the exterior fluid domain. A similar boundary condition can be used to drive a variety of one way interface based fluid/structure or other code-to-code couplings.

Note that a general acoustic source mapping will include both a volume source term and a surface motion boundary condition defined in two variable interpolation blocks.

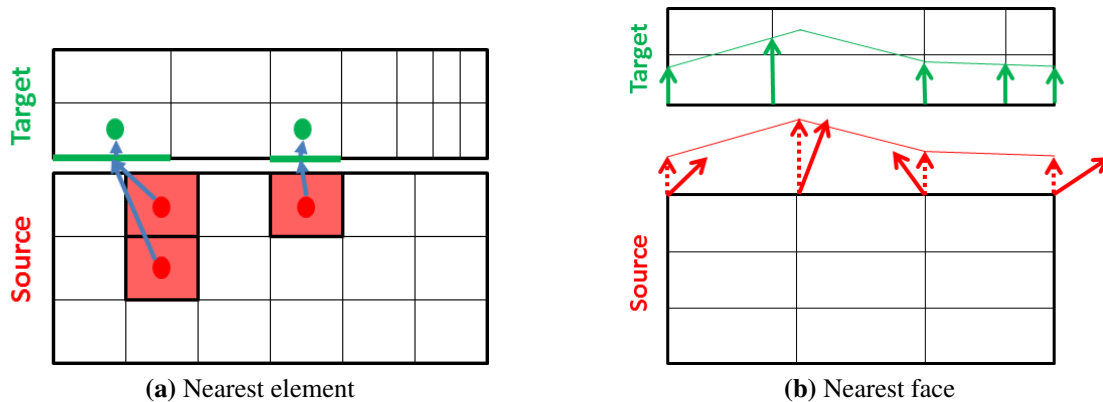


Figure 9.3: Variable interpolation options.

9.9 Global Output Options

These commands exist at the region scope to control the output of default global variables:

```
GLOBAL ENERGY REPORTING = EXACT|APPROXIMATE|OFF (EXACT
EXTENSIVE GLOBAL VARS    PROBLEMSIZE|CORESOLVER|MLSOLVER|CONTACT|
                           ALMATERIAL|CONSTRAINTMESH|ENERGYBALANCE|
                           NODAL|ELEMENT|ALL
                           [IN <output_name: string>]
EXTENSIVE RIGID BODY VARS OUTPUT = OFF|HISTORY|RESULTS|ALL|
                                   DEFAULT (DEFAULT)
```

Through the `GLOBAL ENERGY REPORTING` command line Sierra/SM offers three reporting options for global energy variables: `EXACT`, `APPROXIMATE`, and `OFF`. The `EXACT` and `APPROXIMATE` reporting options use different algorithms for tracking the global values of external energy, internal energy, contact energy, and hourglass energy. In many cases, the `APPROXIMATE` reporting option will provide a modest performance improvement with a negligible effect on the reported energy values. The `OFF` option will result in a further performance improvement and will report all energy values as zero.

The `GLOBAL ENERGY REPORTING` command has no effect on the analysis itself; the energy values calculated are used only for reporting purposes.

The command `EXTENSIVE GLOBAL VARS` requests that a set of global variables (dictated by the choice of the option) be written to the output file(s). An output results file may optionally be specified to which to write the variables; otherwise they will be written to all output files. Note, by default energy and momentum variables are written to all output files. Regardless of the option set here any global variable may be requested specifically by name in an output block.

The line command `EXTENSIVE RIGID BODY VARS OUTPUT` controls the default output set for global rigid body variables. Regardless of the option choice here, global rigid body variables may be output by name in the history or results output blocks. See Table 9.3 for a list of available variables. The `EXTENSIVE RIGID BODY VARS OUTPUT` options are: `OFF` to specify no default rigid body global variable output; `HISTORY` to specify default rigid body global variable output to the history file(s) only; `RESULTS` to specify default rigid body global variable output to the results file(s) only; `ALL` to specify default rigid body global variable output to both the history and results files. This option defaults to `DEFAULT` by default if the analysis contains less than ten rigid bodies then the rigid body variables will be written to results and history. If the analysis has more than ten rigid bodies then no rigid body global variable will be output by default. For an analysis with several rigid bodies it may a significant file size and performance benefit to not output the rigid body variables due to the large number of such variables.

9.10 Variables

This section lists commonly used variables that the user can be output to the results, history, or restart files. This includes global, nodal, and element variables. Material model specific output variables are described in the material model chapter 5.

9.10.1 Global, Nodal, Face, and Element Variables

This section lists commonly used variables. The variables are presented in tables based on use as follows:

- Table 9.1 Energy Variables
- Table 9.2 Other Global Variables for All Analyses
- Table 9.3 Global Variables for Rigid Bodies
- Table 9.4 Nodal Variables for All Analyses
- Table 9.5 Nodal Variables for Implicit Analyses
- Table 9.6 Nodal Variables for Shells
- Table 9.7 Nodal Variables for Spot Welds
- Table 9.8 Nodal Variables for Contact
- Table 9.9 Element Variables for All Elements
- Table 9.10 Element Variables for Solid Elements
- Table 9.11 Element Variables for Membranes
- Table 9.12 Element Variables for Shells
- Table 9.13 Element Variables for Trusses
- Table 9.14 Element Variables for Cohesive Elements
- Table 9.15 Element Variables for Nonlinear Beams
- Table 9.16 Element Variables for Springs
- Table 9.17 Face Variables

The tables provide the following information about each variable:

Variable Name. The variable can be accessed via this name in an results output block, user output block, user subroutine, etc.

Type. This is the intrinsic variable type. The various types are listed as follows:

- `Integer`: A single integer value.
- `Integer[]`: An array of integer values.
- `Real`: A single floating point value.
- `Real[]`: An array of floating point values.
- `Vector_2D`: A two component vector value. Values will be output as `name_x` and `name_y`.
- `Vector_3D`: A three component vector value. Values will be output as `name_x`, `name_y`, and `name_z`.
- `SymTen33`: A symmetric 3x3 tensor with six unique components `name_xx`, `name_yy`, `name_zz`, `name_xy`, `name_yz`, `name_zx`.
- `FullTen36`: A full 3x3 tensor with nine unique components `name_xx`, `name_yy`, `name_zz`, `name_xy`, `name_yz`, `name_zx`, `name_yx`, `name_zy`, and `name_xz`.

Derived. Standard variables are stored in the mesh database. A variable designed with a `yes` in this column is a derived variables. Derived variables are computed on the fly as a function of other variables. For example `unrotated_stress` is a standard variable while `von_mises` is a derived norm of the `unrotated_stress` field. Usually standard and derived variables can be used interchangeably. There are some distinctions. Derived variables cannot be set in initial conditions. Additional derived variables must be included in a `BEGIN DERIVED OUTPUT` command block if it is to be transferred to another procedure or region as described in Section 6.7.

For multi-integration point elements, quantities from the element integration points will be appended with a numerical suffix indicating the integration point. The suffix will range from 1 to the number of integration points. Note, generally if there is only a single integration point no suffix will be present. The suffix is padded with leading zeros. If the number of integration points is less than 10, the suffix has the form `_i`, where *i* ranges from 1 to the number of integration points. If the number of integration points is greater than or equal to 10 and less than 100, the sequence of suffixes takes the form `_01`, `_02`, `_03`, and so forth. Finally, if the number of integration points is greater than or equal to 100, the sequence of suffixes takes the form `_001`, `_002`, `_003`, and so forth. As an example, if `von_mises` is requested for a shell element with 15 integration points, then the quantities `von_mises_01`, `von_mises_02`, ..., `von_mises_15` are output for the shell element.

For node-based tetrahedra the “element” variables are stored at both the elements and at the nodes; more details are available in Section 6.2.1.1. The tables of various types of variables follow.

Table 9.1: Global Energy Variables

Variable Name	Type	Comments
contact_energy	Real	The energy added or subtracted from the system by contact forces. Includes frictional losses and plastic contact. Computed as the integral of <code>contact_force dot velocity</code>
external_energy	Real	The energy added or subtracted from the system by external forces (force, pressure, kinematic BCs, etc.) Computed as integral of <code>force_external dot velocity</code>
internal_energy	Real	Energy stored by material and hourglass forces. Computed as integral of <code>internal+force dot velocity</code>
kinetic_energy	Real	Total kinetic energy of model. Computed as sum of nodal kinetic energies
hourglass_energy	Real	The internal energy due to hourglass forces. Will be present in under-integrated elements such as the uniform gradient hexahedron. <code>hourglass_energy</code> is a component of <code>internal_energy</code>
strain_energy	Real	Total strain energy accumulated in all elements. Integral of stress dot strain. <code>strain_energy</code> is a component of <code>internal_energy</code>
deposited_energy	Real	Total energy deposited by PRESCRIBED ENERGY DEPOSITION boundary conditions. Generally only relevant to equation of state based material models
ke_blockblockID	Real	Kinetic energy sum for each block, i.e., <code>ke_block1</code> for blockID 1
ee_strain_blockblockID	Real	External energy sum for each block, i.e., <code>ee_strain_block1</code> for blockID 1
ie_strain_blockblockID	Real	Internal energy sum for each block, i.e., <code>ie_strain_block1</code> for blockID 1
hge_blockblockID	Real	Hourglass energy sum for each block, i.e., <code>hge_block1</code> for blockID 1

Table 9.2: Other Global Variables For All Analyses

Variable Name	Type	Comments
momentum_blockblockID	Vector_3D	Momentum sum for each block, i.e., momentum_block1 for blockID 1
momentum	Vector_3D	Total model momentum sum. Computed as mass dot velocity
angular_momentum	Vector_3D	Total model angular momentum sum. Includes angular momentum of both the translational and rotational degrees of freedom.
stepcount	Real	Step count (1, 2, ..., N)
timestep	Real	Current time step
timestep_element	Real	Time step from element estimator
timestep_nodal	Real	Time step from nodal estimator
timestep_material	Real	Time step from material model
timestep_lanczos	Real	Time step from Lanczos estimator
timestep_powermethod	Real	Time step from power method estimator
wall_clock_time	Real	Accumulated wall clock run time in seconds
wall_clock_time_per_step	Real	Wall clock time for last time step in seconds
cpu_time	Real	Accumulated CPU time in seconds
cpu_time_per_step	Real	CPU time for last time step in seconds
ghost_count	Real	Total number of off-processor nodes and elements at any given time. If ghost count is growing substantially over time it may indicate that the analysis could be sped up via an automated rebalance.

Table 9.3: Global Variables for Rigid Bodies. (See Section 9.9 for default output options.)

Variable Name	Type	Comments
a<x y z>_<rb_name>	Real	Translational acceleration
vel<x y z>_<rb_name>	Real	Translational velocity
displ<x y z>_<rb__name>	Real	Translational displacement
rota<x y z>_<rb__name>	Real	Rotational acceleration
rotv<x y z>_<rb__name>	Real	Rotational velocity
rotd<x y z>_<rb__name>	Real	Rotational displacement
react<x y z>_<rb_name>	Real	Translational reaction
rreact<x y z>_<rb__name>	Real	Rotational reaction
qvec<1 2 3 4>_<rb__name>	Real	Unit quaternion
Example: ax_rb1 is the x translational acceleration for the rigid body specified in a BEGIN RIGID BODY rb1 block where rb1 is a name chosen by the user.		

Table 9.4: Nodal Variables for All Analyses

Variable Name	Type	Comments
model_coordinates	Vector_3D	Original coordinates of nodes
coordinates	Vector_3D	Current coordinates of nodes
displacement	Vector_3D	Total displacement, generally coordinates minus model_coordinates
velocity	Vector_3D	Velocity of nodes, derivative of displacement
acceleration	Vector_3D	Acceleration of nodes, derivative of velocity
force_internal	Vector_3D	Force produced by element and material response. Also includes hourglass and other element stabilization forces.
force_external	Vector_3D	Force produced by external loading. Force, pressure, as well as reaction forces of kinematic boundary conditions.
force_external_transferred	Vector_3D	Force transferred from another physics (coupled problems only)
force_contact	Vector_3D	Force produced by contact
reaction	Vector_3D	Force produced by kinematic boundary conditions such as fixed displacement
mass	Real	Mass of each node
nodal_time_step	Real	Nodal stable time step (explicit control nodes, coarse mesh only)
node_id	Integer	Global ID of each node
quaternion	Real	Current quaternion (rigid body reference nodes only)
node_id	Int	Exodus global id of the node

**Table 9.5: Nodal Variables for Implicit Analyses**

Variable Name	Type	Comments
displacement_increment	Vector_3D	Change in displacement over the current step
residual	Vector_3D	Nodal free body force imbalance at current time step

Table 9.6: Nodal Variables for Shells and Beams

Variable Name	Type	Comments
rotational_displacement	Vector_3D	Total spin of each node
rotational_velocity	Vector_3D	Spin rate of each node, derivative of rotational_displacement
rotational_acceleration	Vector_3D	Spin acceleration of each node, derivative of rotational_velocity
moment_internal	Vector_3D	Nodal moment from element and material response
moment_external	Vector_3D	Applied external moment, for example from a “PRESCRIBED MOMENT” boundary condition
moment_external_transferred	Vector_3D	Moment transferred from another physics (coupled problems only)
rotational_reaction	Vector_3D	Applied moment to enforce a kinematic boundary condition such as “PRESCRIBED ROTATION”
rotational_mass	Real	Nodal inertia. A single value, nodal inertia assumed spherical.

Table 9.7: Nodal Variables for Spot Welds

Variable Name	Type	Comments
spot_weld_parametric_coordinates	Vector_2D	Parametric coordinates defining node location on face
spot_weld_normal_force_at_death	Real	Value of force normal to face when spot weld breaks
spot_weld_tangential_force_at_death	Real	Value of force tangential to face when spot weld breaks
spot_weld_death_flag	Integer	alive = 0, dead = FAILURE DECAY CYCLES (default is 10), -1 = no spot weld constructed at this node
spot_weld_scale_factor	Real	Nodal influence area of current node
spot_weld_normal_displacement	Real	Current displacement of weld normal to face
spot_weld_tangential_displacement	Real	Current displacement of weld tangential to face
spot_weld_normal_force	Real	Current force of weld normal to face
spot_weld_tangential_force	Real	Current force of weld tangential to face
spot_weld_stiffness	Real	Current stiffness of weld
spot_weld_norm_stiffness	Real	Current stiffness of weld normal to face
spot_weld_tang_stiffness	Real	Current stiffness of weld tangential to face
spot_weld_initial_offset	Vector_3D	The initial offset of the spot weld node from the spot weld surface. Does not change over time, only output if IGNORE INITIAL OFFSET = YES is specified at input.
spot_weld_initial_normal	Vector_3D	The initial normal of the spot weld surface at the point of interaction. Only output if IGNORE INITIAL OFFSET = YES is specified at input.

Table 9.8: Nodal Variables for Contact (See Section 8.10)

Variable	Type	Description
contact_strength	Real	Strength of contact interactions at the node, computed as nodal <code>contact_area</code> divided by <code>surface_area</code> . Nominal range is between 0.0 (no contact) and 1.0 (fully in contact). Values somewhat above 1.0 may be reported in some circumstances. Generally contact strength values between 0.85 and 1.15 will be seen in fully overlapped zones and the contact strength will fade to zero near the edge of contact zones.
contact_status	Real	Status of the interactions at the node. Possible values are as follows: 0.0 = Node is not a contact node (not in a defined contact surface) 0.5 = Node is not in contact For explicit contact: 1.0 = Node is in contact For implicit contact: 1.0 = Node is in contact and is slipping -1.0 = Node is in contact and is sticking (celement).
contact_friction_index	Int	Records what friction model is being used by a node. The index number at the node can be compared to a table printed to the log file to determine what friction model name and type a given node is using. Note, a node in multiple interactions will report the friction model associated with the strongest interaction.
contact_normal_direction	Vector_3D	Direction of the constraint. This is, in general, the normal of the face in the interaction (<code>cdirnor</code>).
force_contact	Vector_3D	Force applied by contact.
contact_tangential_direction	Vector_3D	Direction of the contact tangential force (<code>cdirtan</code>).
contact_normal_force_magnitude	Real	Magnitude of the contact force at the node in the direction normal to the contact face. Magnitude of <code>contact_normal_direction</code> .

continued on next page...

... continued from previous page

Variable	Type	Description
contact_tangential_force_magnitude	Real	Magnitude of the contact force at the node in the plane of the contact face. Magnitude of <code>contact_tangential_direction</code> .
contact_normal_traction_magnitude	Real	Contact traction normal to the contact face. <code>contact_normal_force_magnitude</code> divided by <code>contact_area</code> . If there are multiple interactions for this node, the traction only for the last interaction is given. Negative for compression tractions, positive for tensile tractions.
contact_tangential_traction_magnitude	Real	Traction in the plane of the contact face. <code>contact_traction_force_magnitude</code> divided by <code>contact_area</code> . If there are multiple interactions for this node, the traction only for the last interaction is given. This value is generally negative as it opposes slip.
contact_incremental_slip_magnitude	Real	Magnitude of incremental slip over the current time step (<code>cdtan</code>).
contact_incremental_slip_direction	Vector_3D	Normalized direction of incremental slip over the current time step (<code>cdirisl</code>).
contact_accumulated_slip	Real	Magnitude of tangential slip accumulated over the entire analysis. This is the distance along the slip path, and not the magnitude of <code>contact_accumulated_slip_vector</code> (<code>cstan</code>).
contact_accumulated_slip_vector	Vector_3D	Total accumulated tangential slip over the entire analysis (<code>cdirsl</code>).
contact_frictional_energy	Real	Accumulated amount of frictional energy dissipated over the entire analysis.
contact_frictional_energy_density	Real	Accumulated amount of frictional energy dissipated over the entire analysis, divided by the nodal surface area (<code>cetan</code>).
contact_area	Real	Contact area for the node. This is the tributary area around the node for this interaction. For multiple interactions, the reported area is the area associated with the last interaction (<code>care</code>).
surface_area	Real	Exposed surface area for the node. This is the tributary area around the node.

continued on next page...

... continued from previous page

Variable	Type	Description
contact_normal_gap	Real	Magnitude of gap in the direction normal to the face (cgnor).
contact_tangential_gap	Real	Magnitude of gap in the direction tangent to the face (only applicable for compliant friction models) (cgtan).
removed_overlap	Vector_3D	Direction and magnitude of removed overlap (only applicable for explicit).
contact_energy	Real	Work done by contact on a node by node basis. Positive values indicate contact added energy to the node, negative subtracted energy from the node.

Table 9.9: Element Variables for All Elements. *indicates a derived variable; refer to Section 6.7 for additional details.

Variable	Type	Comments
diagonal_ratio	Real	See Section 2.5
element_mass	Real	Total mass of element
element_id	Integer	Global ID of the element
perimeter_ratio	Real	See Section 2.5
solid_angle	Real	See Section 2.5
timestep	Real	Critical time step for the element. The element in the model with the smallest time step controls the analysis time step.
von_mises*	Real	Von Mises stress norm
hydrostatic_stress*	Real	One-third the trace of the stress tensor
fluid_pressure*	Real	Negative of hydrostatic_stress
stress_invariant_1*	Real	I_1 , Trace of the stress tensor
stress_invariant_2*	Real	I_2 , Second invariant of the stress tensor
stress_invariant_3*	Real	I_3 , Third invariant of the stress tensor
stress_invariant_j2*	Real	J_2 , Second invariant of the deviatoric stress tensor
stress_invariant_j3*	Real	J_3 , Third invariant of the deviatoric stress tensor
triaxiality*	Real	Ratio of hydrostatic pressure to Von Mises stress
lode_angle*	Real	$\frac{1}{3} \arccos(\frac{3}{2} \sqrt{3}(J_3/J_2^{3/2}))$
principal_stresses*	Vector_3D	All three eigenvalues of the stress tensor sorted smallest to largest
max_principal_stress*	Real	Largest eigenvalue of the stress tensor
intermediate_principal_stress*	Real	Middle eigenvalue of the stress tensor
min_principal_stress*	Real	Smallest eigenvalue of the stress tensor
max_shear_stress*	Real	Maximum shear stress from Mohr's circle
octahedral_shear_stress*	Real	Octahedral shear norm of the stress tensor
internal_energy	Real	Total internal energy for the element
hourglass_energy	Real	Total hourglass energy for the element
nonmaterial_strain_3d	SymTen33	Total thermal and artificial strain applied to the element
nonmaterial_strain_rate_3d	SymTen33	Current nonmaterial log strain rate applied to the element
thermal_strain_rate_3d	SymTen33	Current thermal log strain rate applied to the element
artificial_strain_rate_3d	SymTen33	Current artificial log strain rate applied to the element

continued on next page...

... continued from previous page

Variable	Type	Comments
processor_id	Int	Parallel processor rank that owns the element
element_id	Int	Exodus global ID of the element
material_direction_1*	Vector_3D	Material direction 1 for orthotropic materials
material_direction_2*	Vector_3D	Material direction 2 for orthotropic materials
material_direction_3*	Vector_3D	Material direction 3 for orthotropic materials
strain_energy_density	Real	Volumetric material strain energy. Integral of material stress dotted with material strain. Note the <code>strain_energy_density</code> resultant removes the energy associated with thermal strains and artificial bulk viscosity that are included in the <code>deformation_energy_density</code>
strain_energy	Real	Total strain energy, strain energy times element volume

Table 9.10: Element Variables for Solid Elements

Variable	Type	Comments
aspect_ratio	Real	Tetrahedra only. See Section 2.5
dilmod	Real	Dilatational modulus
nodal_jacobian_ratio	Real	Hex and Quad only. See Section 2.5
scaled_jacobian	Real	Hex only. See Section 2.5
element_shape	Real	Element shape quality metric. See Section 2.5
rate_of_deformation	SymTen33	Rate of deformation tensor, $\mathbf{D}$ (Hexahedra and node-based tetrahedra only)
deformation_gradient	FullTen36	Deformation gradient tensor, $\mathbf{F}$
right_stretch	SymTen33	Right stretch tensor, $\mathbf{U}$
left_stretch	SymTen33	Left stretch tensor, $\mathbf{V}$
rotation	FullTen36	Rotation tensor, $\mathbf{R}$
shrmod	Real	Shear modulus
stress	SymTen33	Cauchy stress tensor in global configuration. This is a single volume averaged value for multiple integration point elements.
cauchy_stress	SymTen33	Cauchy stress tensor in the global configuration. This value is present at each integration point.
unrotated_stress	SymTen33	Cauchy stress in the unrotated material configuration. This value is present at each integration point.
max_principal_stress_direction	Vector_3D	Eigenvector of the Cauchy stress tensor corresponding to the largest eigenvalue
intermediate_principal_stress_direction	Vector_3D	Eigenvector of the Cauchy stress tensor corresponding to the middle eigenvalue
min_principal_stress_direction	Vector_3D	Eigenvector of the Cauchy stress tensor corresponding to the smallest eigenvalue
max_principal_strain_direction	Vector_3D	Eigenvector of the strain tensor corresponding to the largest eigenvalue
intermediate_principal_strain_direction	Vector_3D	Eigenvector of the strain tensor corresponding to the middle eigenvalue
min_principal_strain_direction	Vector_3D	Eigenvector of the strain tensor corresponding to the smallest eigenvalue
green_lagrange_strain	SymTen33	Green-Lagrange strain tensor, $\mathbf{E} = (1/2)(\mathbf{F}^T \cdot \mathbf{F} - \mathbf{I})$
biot_strain	SymTen33	Biot Strain tensor, $\mathbf{U} - \mathbf{I}$
log_strain	SymTen33	Log strain tensor, $\ln \mathbf{V}$
unrotated_log_strain	SymTen33	Log strain tensor in unrotated configuration, $\ln \mathbf{U}$

continued on next page...

... continued from previous page

Variable	Type	Comments
effective_log_strain	Real	Effective log strain
log_strain_invariant_1	Real	Trace of the log strain tensor
log_strain_invariant_2	Real	Second invariant of the log strain tensor
log_strain_invariant_3	Real	Third invariant of the log strain tensor
max_principal_log_strain	Real	Largest eigenvalue of the log strain tensor
intermediate_principal_log_strain	Real	Middle eigenvalue of the log strain tensor
min_principal_log_strain	Real	Smallest eigenvalue of the log strain tensor
max_shear_log_strain	Real	Maximum shear log strain from Mohr's circle
octahedral_shear_log_strain	Real	Octahedral strain norm of the log strain tensor
volume	Real	Element volume
internal_energy_density	Real	Element total internal energy divided by current element volume
hourglass_energy_density	Real	Element hourglass energy divided by current element volume
deformation_energy_density	Real	$\text{internal\_energy\_density} - \text{hourglass\_energy\_density}$. Equivalent to the integral of total element stress dotted with total element deformation.
iplocation	Vector_3D	Location of each element integration points in the original model coordinates
centroid	Vector_3D	Location element mass-centroid in the original model coordinates

Table 9.11: Element Variables for Membranes

Variable Name	Type	Comments
memb_stress	SymTen33	Cauchy membrane stress tensor in the global configuration.
element_area	Real	Surface area of the element
element_thickness	Real	Thickness provided to element

Table 9.12: Element Variables for Shells. *indicates a derived variable; consult Section 6.7 for additional details. †indicates a variable which is available for analytic integration; refer to Section 6.2.4.

Variable Name	Type	Comments
memb_stress ^{*,†}	SymTen33	Cauchy stress at mid plane in global X, Y, and Z coordinates.
bottom_stress ^{*,†}	SymTen33	Cauchy stress at bottom integration point in global X, Y, and Z coordinates
top_stress ^{*,†}	SymTen33	Cauchy stress at top integration point in global X, Y, and Z coordinates
unrotated_stress	SymTen33	Cauchy stress in the unrotated material configuration. This value is present at each integration point. ZZ component is undefined.
transform_shell_stress [*]	SymTen21	In-plane shell stress
strain	SymTen33	Integrated strain at mid plane in local shell coordinate system
effective_strain [*]	Real	Effective strain norm
strain_invariant_1 [*]	Real	Trace of the strain tensor
strain_invariant_2 [*]	Real	Second invariant of the strain tensor
strain_invariant_3 [*]	Real	Third invariant of the strain tensor
max_principal_strain [*]	Real	Largest eigenvalue of the strain tensor
intermediate_principal_strain [*]	Real	Middle eigenvalue of the strain tensor
min_principal_strain [*]	Real	Smallest eigenvalue of the strain tensor
max_shear_strain [*]	Real	Maximum shear strain from Mohr's circle
octahedral_shear_strain [*]	Real	Octahedral strain norm of the strain tensor
transform_shell_strain [*]	Real	In-plane shell strain
element_area	Real	Element surface area
element_thickness	Real	Element current thickness
rate_of_deformation	SymTen33	Rate of deformation (stretching) tensor
thickness_to_length_ratio ^{*,†}	Real	Ratio of shell thickness to in-plane element size. Shells with ratios beyond 1.0 may encounter stability issues

Table 9.13: Element Variables for Trusses

Variable Name	Type	Comments
init_length	Real	Initial length of the truss element
truss_strain	Real	Axial strain in the truss element
unrotated_stress	SymTen33	Axial Cauchy stress is stored in unrotated_stress_xx. All other components are zero. See Section 6.2.8 for more details.
truss_force	Real	Axial force in the truss

Table 9.14: Element Variables for Cohesive Elements

Variable Name	Type	Comments
cse_traction	Vector_3D	_z component is normal direction. _x and _y components are tangential directions.
cse_separation	Vector_3D	_z component is normal direction. _x and _y components are tangential directions.
cse_initial_trac	Vector_3D	Available only if traction initialization is used. _z component is normal direction. _x and _y components are tangential directions.
cse_activated	Integer	For intrinsic elements
cse_fracture_area	Real	Currently not used

Table 9.15: Element Variables for Nonlinear Beams

Variable Name	Type	Comments
beam_strain_axial	Real	Axial strain at each integration point.
beam_strain_shear	Real	Shear torsional strain at each integration point.
beam_strain_inc	Vector_2D	Strain increments at each integration point. Axial strain increments are $\_x$ and shear torsional strain increments are $\_y$.
beam_stress_axial	Real	Axial Cauchy stress at each integration point.
beam_stress_shear	Real	Shear circumferential torsional stress at each integration point.
unrotated_stress	SymTen33	Cauchy stresses at each integration point. Only the $\_xx$ and $\_xy$ values per integration point contain actual stress values. Axial stresses are in $\_xx$ and circumferential torsional shear stresses are in $\_xy$. See Section 6.2.7 for more details.
beam_axial_force	Real	Axial force at midpoint.
beam_transverse_force_s	Real	Transverse shear in s -direction at midpoint.
beam_transverse_force_t	Real	Transverse shear in t -direction at midpoint.
beam_moment_r	Real	Torsion at midpoint.
beam_moment_s	Real	Moment about s -direction at midpoint.
beam_moment_t	Real	Moment about t -direction at midpoint.
beam_avg_rate_of_def	Real	Average rate of deformation over all integration point.
current_area	Real	Current area of the beam cross-section.

**Table 9.16: Element Variables for Springs**

Variable Name	Type	Comments
spring_force	Real	Magnitude of the internal spring force.
spring_engineering_strain	Real	Change in length over initial length $\frac{dL}{L_0}$.
spring_init_length	Real	Initial spring length, L_0 .

Table 9.17: Face Variables

Variable Name	Type	Comments
pressure	Real	Pressure applied to faces via a pressure boundary condition

9.10.2 Variables for Material Models

State variables can be output from the material models. Most of the materials, with the exception of simple models such as the elastic model, have state variables. The state variables for material models in LAME are accessible directly by name. For instance, the equivalent plastic strain variable is accessible by the name `EQPS` for all elastic-plastic material models. State variables can also be accessed by index.

Available LAME state variable names for a material will also be listed in the log file from a run that uses the material model.

Tables of available material state variables for commonly used material models are provided in the Materials Chapter [5](#). Note, some models produce slightly different state variable sets for solids vs. shells vs. beams as described in the material model documentation.

9.11 Solution Verification

Solution verification is the capability to enable the code to check a computed solution versus some reference solution and abort the run if an anomaly is detected. Solution verification is primarily used by the code development team to create automated verification tests for various capabilities. Solution verification can be of use to the code end user to help write tests to verify that a user written subroutine or expression is behaving as expected. Solution verification could also be used to stop a run that has entered an unphysical or unneeded regime.

```
BEGIN SOLUTION VERIFICATION
  VERIFY GLOBAL <string>gVar <|<=|=|>|>=
    <real>gVal [PLUS OR MINUS <real>Tol]
  VERIFY GLOBAL <string>gVar <|<=|=|>|>=
    GLOBAL <string>gVar2 [PLUS OR MINUS <real>Tol]
  VERIFY GLOBAL <string>gVar <|<=|=|>|>=
    FUNCTION <string>funcName
  VERIFY NODAL <string>nVar
    AT NODE <integer>nodeNum <|<=|=|>|>=
      FUNCTION <string>funcName
  VERIFY NODAL <string>nVar
    AT NODE <integer>nodeNum <|<=|=|>|>=
      <real>val
  VERIFY ELEMENT <string>eVar
    AT ELEMENT <integer>elemNum <|<=|=|>|>=
      <real>val
  VERIFY ELEMENT <string>eVar
    AT ELEMENT <integer>elemNum <|<=|=|>|>=
      FUNCTION <string>funcName

  # Absolute tolerance
  TOLERANCE = <real>absTol
  # Relative tolerance
  RELATIVE TOLERANCE = <real>relTol
  # Time tolerance
  TIME TOLERANCE = <real>timeTol

  SKIP TIMES = <real>time0 TO <real>time1

  COMPLETION FILE = <string>fileName
END
```

The various forms of the VERIFY GLOBAL, VERIFY NODAL, VERIFY ELEMENT commands check that the named variable has some relation to a defined constant, time history function, or other global variable. For example if creating and debugging a user subroutine based displacement function it may be helpful to ensure that the displacement at some node tracks with the expected analytic displacement given in a time history function. The global variables used in solution verification are often created via a USER OUTPUT based command block. If at any time step at which

the solution is verified the `VERIFY` statement fails, the analysis will immediately abort and report an error.

Tolerances of how close a verification needs to be in order to be acceptable can be specified. The `TOLERANCE` and `PLUS OR MINUS` verify option both specify absolute tolerances. For example the command:

```
VERIFY GLOBAL disp1 = 1.0 PLUS OR MINUS 0.1
```

will verify successfully as long as `disp1` is between 0.9 and 1.1.

The `RELATIVE TOLERANCE` command specifies an acceptable relative error between values being verified. A verification is successful if:

$$\frac{|gVal - gVar|}{\max(|gVal|, |gVar|)} < relTol \quad (9.1)$$

The `TIME TOLERANCE` is used with the time history function based verify commands. A verification will pass if the `gVar` is within tolerance to any value of the function that would be computed at the current time plus or minus the `timeTol`. The time tolerance command is useful for verifying functions that have sharp jumps in time.

The `SKIP TIMES` command is used to skip verification over the defined time periods. The skip times command can be used to focus the verification on only the time periods where the solution is known.

The `COMPLETION FILE` command causes solution verification to output a status file named `fileName` at successful completion of the code run. Existence of the status file indicates that the run completed as expected. If the run dies due to failing a solution verification, runs into a queue time limit, or other unexpected error, the status file will not be written.

9.12 Identify Fragments

The `IDENTIFY FRAGMENTS` capability provides data on unique contiguous mesh fragments in a simulation. Fragments are identified based on mesh connectivity. The fragment identification algorithm accounts for topological changes including element death, element separation in an XFEM simulation, and bond failure in a peridynamics simulation. Fragment identification element field `elem_fragment_id` and nodal field `node_fragment_id` indicate the fragment number for each element and node in the simulation, respectively, and may be output in a results block, allowing for fragment visualization and other post-processing operations. Fragment numbers range from 1 to the total number of fragments, sorted in order of increasing fragment mass. Additional fragment data, including fragment mass and momentum, may be written to an optional output text file.

A nodal vector field `fragment_force` is provided to compute force and moment on a per-fragment basis. If using an applied pressure (Section 7.6.1), the `fragment_force` field can be used as the external force contribution output name. In a Zapotec coupled analysis the field `force_to_apply` might coupled with `fragment_force` to report force and moments on the fragments. Otherwise, `fragment_force` can be populated through user output methods.

Fragment identification is controlled by the `BEGIN IDENTIFY FRAGMENTS` block in the region scope:

```
BEGIN IDENTIFY FRAGMENTS
  OUTPUT FILE = <string>output_filename
  INCREMENT = <real>increment
  MINIMUM FRAGMENT SIZE = <int>min_element_threshold
END [IDENTIFY FRAGMENTS]
```

The optional `OUTPUT FILE` command triggers output to a text file named `output_filename` containing fragment data. The data written to the text file include the mass, momentum, center of mass, moment of inertia, angular momentum, and names of element blocks present in each fragment. The optional `MINIMUM FRAGMENT SIZE` command groups fragments containing fewer than `min_element_threshold` elements into a single fragment with ID 0. This feature is useful, for example, in a brittle fracture simulation that results in a significant number of isolated elements. By default, the fragment identification algorithm is executed once per output step. If the optional `INCREMENT` command is included in the `IDENTIFY FRAGMENTS` block, the fragment identification algorithm is executed only at the corresponding simulation times. For example, if `INCREMENT` is assigned a value of 0.23, the fragment identification algorithm will be executed only when the simulation time reaches a multiple of 0.23. Limiting the number of times the fragment identification algorithm is executed reduces the associated computational expense.



Warning: Fragment identification is performed based only on node-to-element connectivity. If two elements share a node, those two elements are in the same fragment. Other mechanisms to connect elements, such as tied contact, MPCs, spot welds, etc., are not considered in fragment identification at this time. In addition, peridynamics is the only particle method currently supported by fragment identification.

9.13 Detect Breach

The breach detection capability enables automatic detection of a breach creating an open path between two user-defined surfaces via element. This capability can be used to predict when venting of a pressure vessel begins due to an open crack. The `BEGIN DETECT BREACH` block is defined in the region scope with the following command block:

```
BEGIN DETECT BREACH <string>breachName
  SURFACE1 = <string list>surface1_names
  SURFACE2 = <string list>surface2_names
END [DETECT BREACH]
```

The `breachName` string is required, as it defines a global variable with the same name that initializes to 0 and changes to 1 when breach is detected. Within the `DETECT BREACH` block, two line command are required: `surface1 = sideset1 sideset2 ...` and `surface2 = sideset3 sideset4 ...`. These command lines set the “inner” and “outer” surfaces for detecting breach between. An arbitrary number of detect breach blocks are allowed.

The algorithm is implemented by identifying clusters of dead elements. If two dead elements share a node, then they are considered to be in the same cluster. (An alternative implementation could consider dead elements to be in the same cluster only when they share a face; however, that is not the version currently implemented.) If a cluster of dead elements has an adjacent face in both `surface1` and `surface2`, then we say breach has occurred between those surfaces(see Figure 9.4).

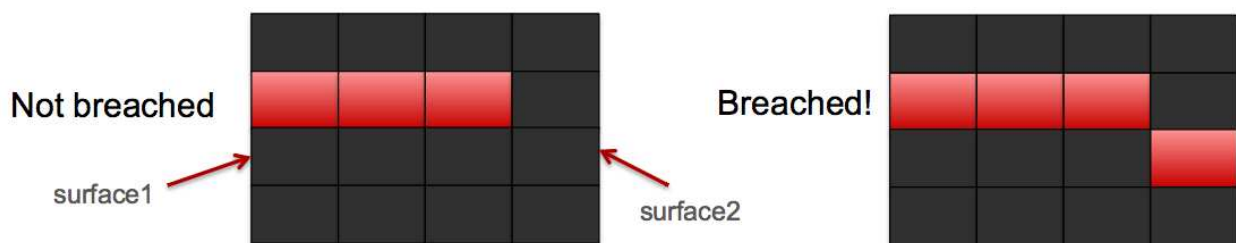


Figure 9.4: Breach detection between surfaces 1 and 2. Dead elements in red.

The death cluster id number can be output as either an element or nodal variable named `dead_element_cluster_id`. The nodal variable version is only used for implementing the clustering algorithm, but the element variable `dead_element_cluster_id` can be output and used by analyst scripts to compute quantities of interest for the dead element clusters such as the volume, minimum cross-sectional area, moments of inertia, etc.



Warning: This capability is currently only tested for bulk 3D elements such as hexahedra and tetrahedra. The capability will not work with shells. Additionally, the capability has no knowledge of tied contacts; thus a breach can only propagate between mesh-contiguous elements.

References

- [1] L.A. Schoof and V.R. Yarberry. EXODUS II: A finite element data model. Technical Report SAND92-2137, Sandia National Laboratories, Albuquerque, NM, September 1994. [pdf](#). See also documentation available at EXODUS II Sourceforge page. [link](#).
- [2] The eXtensible Data Model and Format (xdmf). [link](#).
- [3] G.D. Sjaardema. APREPRO: An algebraic preprocessor for parameterizing finite element analyses. Technical Report SAND92-2291, Sandia National Laboratories, Albuquerque, NM, 1999. [pdf](#).
- [4] Sierra/SM Team. Sierra/SM Theory Manual. Technical Report SAND2013-4615, Sandia National Laboratory, Albuquerque, NM, 2013.
- [5] A. Mota, W. Sun, J.T. Ostien, J.W. Foulk III, and K.N. Long. Lie-group interpolation and variational recovery for internal variables. *Computational Mechanics*, 52(6):1281–1299, 2013.

Chapter 10

Peridynamics

This chapter describes the peridynamics functionality available in Sierra/SM. Peridynamics is a non-local formulation of continuum mechanics that is well-suited for modeling material discontinuities such as cracks.



10.1 Peridynamics

Peridynamics is an extension of classical solid mechanics that allows for the modeling of bodies in which discontinuities occur spontaneously [1, 2, 3, 4, 5]. Unlike the classical continuum formulation, the peridynamic expression for the balance of linear momentum does not contain spatial derivatives and instead is based on an integral equation. For this reason, peridynamics is well suited for modeling phenomena involving spatial discontinuities such as cracking. This section presents a basic overview of the peridynamics formulation implemented in Sierra/SM and suggested guidelines for setting up peridynamic analyses. For a comprehensive review of peridynamics theory, the reader is referred to [5].

10.1.1 Overview

Peridynamics is a non-local theory in which any point in the body $\mathbf{x}$ is acted on by forces due to the deformation of all points $\mathbf{x}'$ within a neighborhood of radius δ centered at $\mathbf{x}$. The radius δ is called the horizon and the vector $\mathbf{x}' - \mathbf{x}$ is called a bond.

The peridynamic equation of motion is

$$\begin{aligned}\rho(\mathbf{x})\ddot{\mathbf{u}}(\mathbf{x}, t) &= \mathbf{L}_{\mathbf{u}}(\mathbf{x}, t) + \mathbf{b}(\mathbf{x}, t) \quad \forall \mathbf{x} \in \mathcal{B}, \quad t \geq 0, \\ \mathbf{L}_{\mathbf{u}}(\mathbf{x}, t) &= \int_{\mathcal{B}} \{ \mathbf{T}[\mathbf{x}, t] \langle \mathbf{x}' - \mathbf{x} \rangle - \mathbf{T}[\mathbf{x}', t] \langle \mathbf{x} - \mathbf{x}' \rangle \} dV_{\mathbf{x}'}.\end{aligned}\tag{10.1}$$

Here, $\mathcal{B}$ is the reference configuration of the body, ρ is the density in the reference configuration, $\mathbf{u}$ is the displacement, and $\mathbf{b}$ is the body force density.

The relationship between $\mathbf{x}$ and $\mathbf{x}'$ is expressed in terms of the *force state* at $\mathbf{x}$ at time t , $\mathbf{T}[\mathbf{x}, t]$. The force state is a function that associates with any bond $\mathbf{x}' - \mathbf{x}$ a force density per unit volume $\mathbf{T}[\mathbf{x}, t] \langle \mathbf{x}' - \mathbf{x} \rangle$ acting on $\mathbf{x}$. This force density per unit volume is called a *pairwise force*.

The body $\mathcal{B}$ may be discretized in the reference configuration into a finite number of cells with each cell containing a single node at its center. The integral in Equation (10.1) may then be replaced

by a summation,

$$\rho(\mathbf{x})\ddot{\mathbf{u}}_h(\mathbf{x}, t) = \sum_{i=0}^N \left\{ \underline{\mathbf{T}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle - \underline{\mathbf{T}}'[\mathbf{x}'_i, t] \langle \mathbf{x} - \mathbf{x}'_i \rangle \right\} \Delta V_{\mathbf{x}'_i} + \mathbf{b}(\mathbf{x}, t), \quad (10.2)$$

where N is the number of cells in the neighborhood of $\mathbf{x}$, $\mathbf{x}'_i$ is the position of the node centered in cell i , and $\Delta V_{\mathbf{x}'_i}$ is the volume of cell i . The length of the bond between two cells is determined as the distance between the nodes at the centers of the cells.

Equation (10.2) requires evaluation of the pairwise forces $\underline{\mathbf{T}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle$ and $\underline{\mathbf{T}}'[\mathbf{x}'_i, t] \langle \mathbf{x} - \mathbf{x}'_i \rangle$, where the force states $\underline{\mathbf{T}}[\mathbf{x}, t]$ and $\underline{\mathbf{T}}'[\mathbf{x}'_i, t]$ are, in general, functions of the deformations of all cells within the neighborhoods of $\mathbf{x}$ and $\mathbf{x}'_i$, respectively. A constitutive model is required to compute the force state, $\underline{\mathbf{T}}[\mathbf{x}, t]$, in terms of the deformation state in the neighborhood of $\mathbf{x}$ and possibly other variables as well.

The constitutive models available for peridynamics in Sierra/SM fall into three categories:

1. State-based constitutive models developed specifically for peridynamics.
2. Bond-based constitutive models developed specifically for peridynamics.
3. Classical constitutive models within the LAME library that have been made available from within peridynamics via a wrapper.

Constitutive models developed specifically for peridynamics are generally more robust than classical constitutive models that have been adapted for use within peridynamics, particularly in analyses involving extreme deformation and pervasive fracture. The *Linear Peridynamic Solid* material model developed by Silling, et al. [3], described below, is the only model of the first type currently available in Sierra/SM. Two bond-based constitutive models are available, the *Microelastic* material model [1], and the *Microplastic* material model [6]. The peridynamics interface to classical material models offers a means for accessing the large number of material models within LAME. This interface is based on the computation of an approximate deformation gradient, $\bar{\mathbf{F}}$, described in Section 10.1.5.

10.1.2 State-Based Linear Peridynamic Solid Material Model

The state-based linear peridynamic solid material model computes the pairwise forces $\underline{\mathbf{T}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle$ acting on $\mathbf{x}$ based on the deformation of all cells $\mathbf{x}'$ within the neighborhood of $\mathbf{x}$,

$$\underline{\mathbf{T}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle = \underline{t} \underline{\mathbf{M}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle, \quad (10.3)$$

where $\underline{t}$ is the magnitude of the pairwise force and $\underline{\mathbf{M}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle$ is the unit vector pointing from the deformed position of $\mathbf{x}$ to the deformed position of $\mathbf{x}'$.

The magnitude of the pairwise force in a linear peridynamic solid is given by

$$\underline{t} = \frac{-3p}{m} \underline{\omega} \underline{x} + \frac{15\mu}{m} \underline{\omega} \underline{e}^d, \quad p = -k\theta, \quad (10.4)$$

where the p is the peridynamic pressure, θ is the peridynamic dilatation, m is the weighted volume at $\mathbf{x}$, $\underline{\omega}$ is the value of the influence function between cells $\mathbf{x}$ and $\mathbf{x}'$, $\underline{x}$ is the length of the bond

$\mathbf{x}' - \mathbf{x}$ in the undeformed configuration, $\underline{e}^d$ is the deviatoric part of the extension $\underline{e}$, and μ and k are material constants (the shear modulus and bulk modulus, respectively). For a complete description of the linear peridynamic solid material model, see [3].

10.1.3 Bond-Based Microelastic Peridynamic Material Model

The bond-based microelastic material model computes the pairwise forces $\underline{\mathbf{T}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle$ acting on $\mathbf{x}$ in accordance with Equation (10.3). Unlike the state-based linear peridynamic solid material model, the pairwise force acting between cells $\mathbf{x}$ and $\mathbf{x}'$ is purely a function of their relative displacements. This distinction differentiates *bond-based* peridynamic material models from the more general *state-based* peridynamic material models.

The magnitude of a pairwise force for the microelastic material model is computed as

$$\underline{t} = \frac{18k}{\pi\delta^4} s, \quad (10.5)$$

where k is the bulk modulus, δ is the peridynamic horizon, and s is the bond stretch, defined as

$$s = \frac{\underline{y} - \underline{x}}{\underline{x}}. \quad (10.6)$$

Here, $\underline{x}$ is the length of the bond $\mathbf{x}' - \mathbf{x}$ in the undeformed configuration, and $\underline{y}$ is the bond length in the current configuration. For a complete description of the bond-based microelastic material model, see [1, 2].

10.1.4 Bond-Based Microplastic Peridynamic Material Model

The bond-based microplastic material model computes the pairwise forces $\underline{\mathbf{T}}[\mathbf{x}, t] \langle \mathbf{x}'_i - \mathbf{x} \rangle$ as a function of the relative displacements of the cells $\mathbf{x}$ and $\mathbf{x}'$. As with the microelastic material model, each pairwise force is purely a function of the stretch of an individual bond. In the case of the microplastic material model, an elastic-perfectly-plastic constitutive law is used to compute the pairwise force as a function of bond stretch. The magnitude of a pairwise force for the microplastic material model is computed as

$$\underline{t} = \frac{18k}{\pi\delta^4} (s - s_p), \quad (10.7)$$

where k is the bulk modulus, δ is the peridynamic horizon, s is the total bond stretch, and s_p is the plastic bond stretch. Plastic bond stretch is governed by

$$s_p(0) = 0, \quad \dot{s}_p = \begin{cases} \dot{s} & \text{if } |s - s_p| \geq s_Y, \\ 0 & \text{otherwise.} \end{cases} \quad (10.8)$$

For a complete description of the bond-based microplastic material model, see [6].

10.1.5 Interface to Classical Material Models

A means for adapting classical material models for use with peridynamics has been developed by Silling, et al. [3]. The approach is based on the evaluation of an approximate deformation gradient,

$\bar{\mathbf{F}}$, at $\mathbf{x}$,

$$\bar{\mathbf{F}} = \left(\sum_{i=0}^N \underline{\omega}_i \underline{\mathbf{Y}}_i \otimes \underline{\mathbf{X}}_i \Delta V_{\mathbf{x}_i} \right) \mathbf{K}^{-1}, \quad (10.9)$$

where $\mathbf{K}$ is the *shape tensor*, defined as

$$\mathbf{K} = \sum_{i=0}^N \underline{\omega}_i \underline{\mathbf{X}}_i \otimes \underline{\mathbf{X}}_i \Delta V_{\mathbf{x}_i}. \quad (10.10)$$

Here, $\mathbf{X}$ denotes a vector directed from $\mathbf{x}$ to $\mathbf{x}'$ in the reference configuration, $\mathbf{Y}$ denotes a vector directed from $\mathbf{x}$ to $\mathbf{x}'$ in the deformed configuration, $\underline{\omega}$ is the value of the influence function between cells $\mathbf{x}$ and $\mathbf{x}'$, and $\Delta V_{\mathbf{x}}$ is the volume of cell $\mathbf{x}$. The operator $\otimes$ denotes a dyadic product.

The approximate deformation gradient, as defined in Equation (10.9), allows for the computation of a strain measure or, alternatively, a strain increment, that can be passed to a classical material model. The classical material model is expected to return stress which is then transformed into a pairwise force as follows,

$$\underline{\mathbf{T}} \langle \mathbf{x}' - \mathbf{x} \rangle = f_d \underline{\omega} \boldsymbol{\sigma} \mathbf{K}^{-1} \langle \mathbf{x}' - \mathbf{x} \rangle, \quad (10.11)$$

where $\boldsymbol{\sigma}$ is the Piola stress and f_d is the stress decay function. As described in Section 6.2.9, the stress decay function is a multiplier that reduces the stress as a function of the percentage of broken bonds for a given element. This is necessary to avoid a nonphysical increase in the magnitude of pairwise forces as damage accumulates.

The accuracy of the peridynamic interface to classical material models depends strongly on the calculation of the approximate deformation gradient, $\bar{\mathbf{F}}$. The approximate deformation gradient calculation may become unreliable in cases of extreme deformation, or in cases of pervasive material failure in which a large percentage of bonds at a given point are broken. This may result in unphysical strain values being passed to the classical material model, and a subsequent unphysical internal force calculation. In addition, the inversion of the shape tensor in Equation (10.11) will fail if the number of bonds for a given cell is less than three, or if all bonds for a given cell are coplanar. For these reasons, Sierra/SM sets the stress to zero for peridynamic cells with less than three intact bonds that are not coplanar. Under these conditions, a cell will remain active in the simulation and may be acted on by other cells but will not contribute to the internal force. Cells for which all bonds have been broken will behave as mass particles.

Zero-energy modes are possible in the calculation of the approximate deformation gradient, $\bar{\mathbf{F}}$. Zero energy modes may result in unphysical motion of cells within a simulation, typically manifesting as rapid oscillations. Sierra/SM provides a means for suppressing zero-energy modes through peridynamic hourglass control. Peridynamic hourglass control is based on the comparison of cell positions with the positions of those cells as predicted by direct application of the approximate deformation gradient computed at a cell $\mathbf{x}$. The suppression of zero-energy modes is achieved by applying a penalty force proportional to the difference between these values.

10.1.6 Peridynamic Influence Function

State-based peridynamic material models make use of an influence function, ω , to assign a weighting to each neighbor $\mathbf{x}'$ of a given cell $\mathbf{x}$. An influence function is used for both the Linear Peridynamic Solid material model (Section 10.1.2) and classical material models made available for

peridynamics via a wrapper (Section 10.1.5). Influence functions do not apply to bond-based peridynamics. The default influence function, shown in Figure 10.1, has a value of one for neighbors within a distance of half the horizon, beyond which it drops off parabolically to a value of zero at the horizon. This form of the influence function mitigates numerical noise that can result from cells that lie very near the edge of the horizon. As described in Section 6.2.9, a user-defined influence function may be used in place of the default influence function.

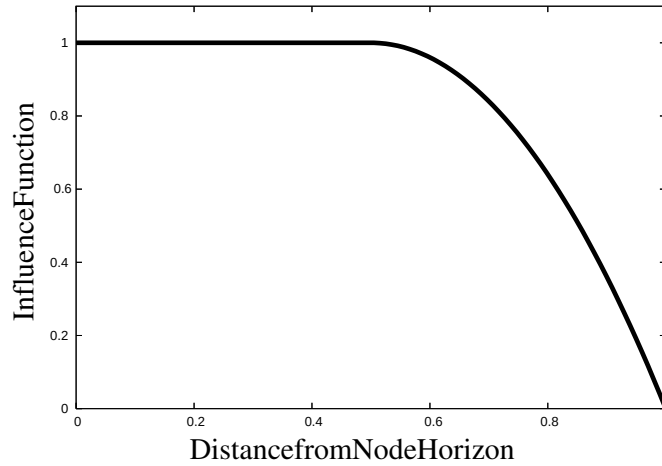


Figure 10.1: Default influence function for peridynamics.

10.1.7 Modeling Fracture

Peridynamics includes a natural mechanism for modeling fracture through the breaking of peridynamic bonds. In an undamaged material, bonds connect a given cell $\mathbf{x}$ to each cell $\mathbf{x}'$ within its neighborhood in the reference configuration. A damage law dictates the conditions under which individual bonds are broken. Material discontinuities, such as cracks, form as a result of the accumulation of broken bonds.

A critical stretch damage model has been implemented in Sierra/SM to allow for the modeling of fracture within a peridynamics simulation. The critical stretch model assigns a damage value of zero (unbroken) or one (broken) as a function of the maximum stretch obtained by a bond and a critical stretch material parameter, s_{crit} . The damage value, ϕ , is given by the relation

$$\phi = \begin{cases} 0 & \text{for } s_{\text{max}} < s_{\text{crit}} \\ 1 & \text{for } s_{\text{max}} \geq s_{\text{crit}} \end{cases} \quad (10.12)$$

where s is the maximum stretch,

$$s_{\text{max}} = \frac{y_{\text{max}} - x}{x}. \quad (10.13)$$

Here, y_{max} is the maximum distance between the bonded cells $\mathbf{x}$ and $\mathbf{x}'$ achieved over the duration of the simulation, and x is the distances between $\mathbf{x}$ and $\mathbf{x}'$ in the reference configuration. The use of the maximum distance, y_{max} , ensures that the breaking of bonds is irreversible.

The critical stretch decay bond breaking law is a modification of the critical stretch bond breaking law in which the critical stretch decays as a function of peridynamic damage as follows:

$$s_{\text{crit}} = s_{\text{crit}_0} \max(m, 1 - c d_{\text{max}}), \quad (10.14)$$

where s_{crit} is the critical stretch, s_{crit_0} is the nominal critical stretch, m is the minimum allowable multiplier, c is the damage coefficient, and d_{max} is the maximum peridynamic damage (percentage of broken bonds) for the two cells connected by the given bond.

The critical stretch pervasive damage bond breaking law is a modification of the critical stretch bond breaking law in which the critical stretch increases as a function of peridynamic damage:

$$s_{\text{crit}} = s_{\text{crit}_0} \min\left(C3, 1 + C2 \frac{d_{\text{max}} - C1}{1 - d_{\text{max}}}\right), \quad (10.15)$$

where $C1$, $C2$, and $C3$ are user-supplied coefficients. The critical stretch pervasive damage law is appropriate for cases in which multiple cracks may pass through the neighborhood of a given cell $\mathbf{x}$.

Sierra/SM also allows for the breaking of peridynamic bonds based on the value of an element variable. A common use case for this option is the breaking of bonds based on the value of a material model variable, such as equivalent plastic strain. In the case where a bond connects two peridynamic cells for which the specified element variable is defined, the bond is broken when the average value of the element variable exceeds the given threshold. In the case where the element variable is defined on only one of the cells, the single available value is compared against the threshold value.

10.1.8 Peridynamics and Contact

Peridynamics elements are compatible with the Dash contact algorithm as described in Chapter 8. The Dash contact algorithm allows for general and self contact for peridynamics blocks and general contact between peridynamics blocks and blocks modeled with classical finite elements. Contact for peridynamics is enabled by the use of lofted contact geometry, which provides the planar facets required by the contact algorithm (see Section 8.9.1.2). In the case of self contact, the contact algorithm has been specialized for peridynamics. Self contact is enabled between peridynamic elements that are not bonded to each other, but is disabled between any pair of elements that is bonded. In this way, peridynamic elements may interact via their constitutive laws, or via contact, but not via both their constitutive laws and contact simultaneously. Self contact between a pair of bonded elements is enabled if that the bond is broken due to material damage.

10.1.9 Usage Guidelines

This section provides basic usage guidelines for carrying out peridynamic analyses with Sierra/SM. Suggestions are provided for constructing input mesh files, applying boundary conditions, and selecting appropriate values for peridynamic parameters. For a description of the PERIDYNAMICS SECTION input syntax, see Section 6.2.9.

Model generation. There are two options for generating a peridynamic model: pre-processing a mesh using `shperegen.py` or reading in a mesh and converting to particles at initialization.

The recommended way to create a peridynamic model is to read in a standard hexahedron and/or tetrahedron element mesh and convert the elements to particles at initialization in the region scope using the `BEGIN CONVERSION TO PARTICLES AT INITIALIZATION` (see also Section 6.2.12):

```
BEGIN CONVERSION TO PARTICLES AT INITIALIZATION
  BLOCK = <string list>block_names
  SECTION = <string>particle_section
END [CONVERSION TO PARTICLES AT INITIALIZATION]
```

where the `BLOCK` command specifies which blocks are converted to particles, and `SECTION` specifies the peridynamics section.

An alternative way to convert a standard mesh is to use the utility `spheregen.py`, which is available for the construction of input meshes that are compatible with the peridynamic capabilities of Sierra/SM. `spheregen.py` converts hexahedron and tetrahedron elements into sphere elements. By default, all hexahedron and tetrahedron blocks are converted to sphere elements. In the case of analyses using both peridynamics and classical finite element analysis, specific blocks may be selected for conversion to sphere elements using the `-b` and `+b` options. A complete list of `spheregen.py` options is available by running `spheregen.py --help`. The `spheregen.py` utility attempts to preserve node sets defined in the original mesh file by transferring them to newly-created sphere elements. Side sets defined in the original mesh file are lost for elements that are converted to sphere elements.

Horizon. The horizon defines the region of nonlocality for a given cell in a peridynamics analysis. Analysis results may be strongly dependent on the choice of the horizon. For example, the choice of horizon affects the localization of forces in the neighborhood of a discontinuity, as well as the dispersion of waves traveling through blocks modeled with peridynamics. In addition to affecting material behavior, the choice of horizon has a strong effect on the computational cost of a peridynamic analysis. In extreme cases, a large horizon may render an analysis intractable. A recommend value of the horizon is three times the mesh spacing. When using the `SCALE BY ELEMENT RADIUS` option, this corresponds to a horizon value of approximately five.

Boundary conditions. Boundary conditions for a nonlocal approach such as peridynamics differ fundamentally from boundary conditions in classical mechanics. To capture exactly the material response in the vicinity of a boundary condition, the boundary condition in a nonlocal model must be applied over a volume. For example, displacement boundary conditions applied at the surface of a body should be applied to a volume that reaches into the body a distance equal to the peridynamic horizon. This is not always practical (as with contact), and often satisfactory results can be obtained by applying boundary conditions at the surface only. Users should be aware, however, that unphysical edge effects will be present if boundary conditions are not applied in a way that is consistent with the nonlocal nature of peridynamics.

Timestep. The time step computed by peridynamic elements in Sierra/SM is an approximate value that may require modification to ensure a stable analysis. Peridynamic elements in Sierra/SM compute a stable time step under an assumed relationship between the bulk modulus in *state-based* peridynamics and the micro-modulus in *bond-based* peridynamics. For details regarding the time step computed by peridynamic elements in Sierra/SM, see [2]. Because the time step

computed by peridynamic elements is not guaranteed to be stable, it is recommended that the time step be reduced using a scale factor of approximately 0.75. This can be achieved using the `TIME STEP SCALE FACTOR` command described in Chapter 3.

Hourglass stiffness. The use of classical material models with peridynamics in Sierra/SM may require the application of hourglass stiffness to reduce the effects of zero-energy modes in the computation of the approximate deformation gradient, $\bar{\mathbf{F}}$. Recommended values for hourglass stiffness are in the range 0.01 - 0.05. The value of hourglass stiffness is multiplied by a material's bulk modulus when computing the stiffening forces for peridynamics hourglass control. For a description of the corresponding input syntax, see Section 6.2.9.

References

- [1] S.A. Silling. Reformulation of elasticity theory for discontinuities and long-range forces. *Journal of the Mechanics and Physics of Solids*, 42:175–209, 2000. [doi](#).
- [2] S.A. Silling and E. Askari. A meshfree method based on the peridynamic model of solid mechanics. *Computers and Structures*, 83:1526–1535, 2005. [doi](#).
- [3] S.A. Silling, M. Epton, O. Weckner, J. Xu, and E. Askari. Peridynamic states and constitutive modeling. *Journal of Elasticity*, 88:151–184, 2007. [doi](#).
- [4] S.A. Silling and R. B. R.B. Lehoucq. Convergence of peridynamics to classical elasticity theory. *Journal of Elasticity*, 93:13–37, 2008. [doi](#).
- [5] S.A. Silling and R.B. Lehoucq. Peridynamic theory of solid mechanics. *Advances in Applied Mechanics*, 44:73–168, 2010. [doi](#). Also available as SAND2010-1233J. [pdf](#).
- [6] R.W. Macek and S.A. Silling. Peridynamics via finite element analysis. *Finite Elements in Analysis and Design*, 43:1169–1178, 2007. [doi](#).

Chapter 11

User Subroutines

User-defined subroutines is a functionality shared by Adagio and Presto. This chapter discusses when and how to use user-defined subroutines. There are examples of user-defined subroutines in the latter part of this chapter. Some of the examples are specific to a given analysis type, i.e., they are applicable to Presto rather than Adagio or vice versa. All examples, regardless of their applicability, do provide important information about how to use the command options available for user-defined subroutines.

In the introductory part of Chapter 11, we first describe, in general, possible applications for the user subroutine functionality in Sierra/SM. Then, again in general, we describe the various pieces and steps that are required by the user to implement a user subroutine. Subsequently, we focus on various aspects of implementing the user subroutine functionality. Section 11.1 describes the details of the user subroutine. Section 11.2 describes the command lines associated with user subroutines that appear in the Sierra/SM input file. In Section 11.3, we explain how to build and use a version of Sierra/SM that incorporates your user subroutine. Finally, Section 11.4 provides examples of actual user subroutines, and Section 11.5 lists some subroutines that are now in the standard user library.

Applications. User subroutines are primarily intended as complex function evaluators that are to be used in conjunction with an existing Sierra/SM capability (boundary conditions, element death, user output, etc.). Some Examples of possible uses of user subroutines are as follows:

- Element death determined by a complex function based on a set of physical parameters and element stress.
- Complex time and space history dependency for a displacement boundary condition.
- Computation the total contact force acting on a given surface and then use of that force to compute a specialized post processing output.
- Transformation of element stress to a arbitrary user defined local coordinate system.
- Computation of cavity volume of an enclosed sideset.
- Aerodynamic pressure based on velocity and surface normal applied to a specified surface.

Some capability exists for using mesh connectivity. It is possible to compute an element quantity based on values at the element nodes.

Some difficulties might occur in parallel applications. If computations for element A depend on quantities in element B and elements A and B are on different processors, then the computations

for A may not have access to quantities in element B. For most computations in user subroutines, however, this should not be a problem.

Implementing completely new capabilities, particularly if these capabilities involve parallel computing, may be difficult or impossible with user subroutines.

General Pieces and Steps. A number of pieces and steps are required to make use of user subroutines. Here, we present a brief description of the pieces and steps that a user will need for user subroutines without going into detail. The details are discussed in later parts of this chapter.

1. You must first determine whether your application fits in the user subroutine format. This can be done by considering the above requirements and examining the description of commands for functionality in Sierra/SM. For example, the basic kinematic boundary conditions and force conditions allow for the use of user subroutines. The description of these commands includes a discussion of how a user subroutine could be applied and what command line will invoke a user subroutine.
2. If you determine that your application can make use of the user subroutine functionality in Sierra/SM, you will then need to write the subroutine. The parts of the subroutine that interface to Sierra/SM have specified formats. The details of these interfaces are described in later sections. One part of the subroutine with a specified format is the call list. Other parts of the subroutine with a specified format are code that will do the following:
 - Read parameters from the Sierra/SM input file
 - Access a variety of information—field variables, analysis time, etc.—from Sierra/SM
 - Store computed quantities

Parameters are values they may be passed from the Sierra/SM input file to the user subroutine. Suppose that the spatial variation for some quantity in the user subroutine uses some characteristic length and the user wishes to examine results generated by using several different values of the characteristic length. By setting up the characteristic length as a parameter, the value for the parameter in the user subroutine can easily be changed by changing the value for the parameter in the input file. This lets the user change the value for a variable inside the user subroutine without having to recompile the user subroutine.

The portion of your subroutine not built on the Sierra/SM specifications will reflect your specific application. The code to implement your application may include a loop over nodes that prescribes a displacement based on the current time for the analysis and the spatial location of the node.

3. After you write the user subroutine, you will need to have a command line in your input file that tells Sierra/SM you want to use the user subroutine you have written. For example, if your user subroutine is a specialized prescribed displacement boundary condition, then inside a `PRESCRIBED DISPLACEMENT` command block, you will have a command line of the form

```
NODE SET SUBROUTINE = <string>subroutine_name
```

that provides the name of your user subroutine.

4. Following the invocation of the user subroutine, there may be command lines for various parameters associated with the user subroutine. There may also be some additional command lines in other sections of the code required for your application. For example, you may have to add command lines in the region scope that will create an internal variable associated with a computed quantity so that the computed quantity can be written to the results file.
5. Once you have constructed the user subroutine, which is a FORTRAN file, and the Sierra/SM input file, you can build an executable version of Sierra/SM that will run your user subroutine. Your Sierra/SM run will then incorporate the functionality you have created in your user subroutine.

Figure 11.1 presents a high-level overview of the various components that work together to implement the user subroutine functionality. The two main components needed for user subroutines, which are commands in the Sierra/SM input file and the actual user subroutine, are represented by the two columns in Figure 11.1.

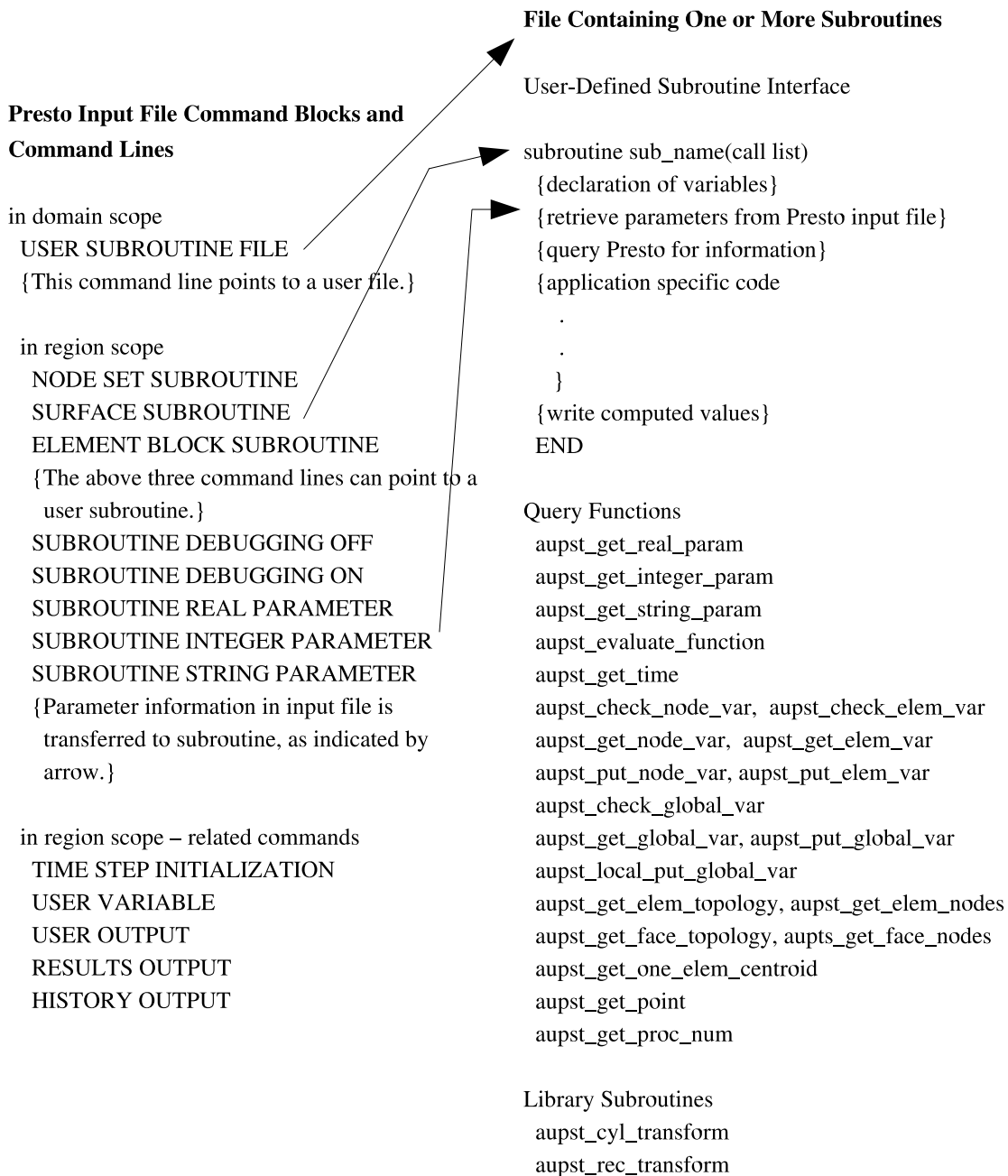


Figure 11.1: Overview of components required to implement user subroutine functionality, excluding compilation and execution commands.

11.1 Programming

Currently, user subroutines are only supported in FORTRAN 77. Any subroutine that can be compiled with a FORTRAN 77 compiler on the target execution machine can be used. The user should be aware that some computers support different FORTRAN language extensions than others. (In the future, other languages such as FORTRAN 90, C, and C++ may be supported.)

User subroutine variable types must interface directly with the matching variable types used in the main Sierra/SM code. Thus, the FORTRAN 77 subroutines should use only integer, double precision, or character types for any data used in the interface or in any query function. Using the wrong data type may yield unpredictable results. The methods used to pass character types from Sierra/SM to FORTRAN user subroutines can be machine-dependent, but generally this functionality works well.

The basic structure for the user subroutine is as follows:

```
subroutine sub_name(call list)
{declaration of variables}
{retrieve parameters from Sierra/SM input file}
{query Sierra/SM for information}
{application-specific code
.
.
}
{write computed values}
END
```

In general, the user will begin the subroutine with variable declarations. After the variable declarations, the user can then query the Sierra/SM input file for parameters. Additional Sierra/SM information such as field variables or element topology can then be retrieved from Sierra/SM. Once the user has collected all the information for the application, the application-specific portion of the code can be written. After the application-specific code is complete, the user may store computed values.

Section 11.1.1 through Section 11.1.3 describe in detail the format for the interfaces to Sierra/SM that will allow the user to make the subroutine call, retrieve information from Sierra/SM, and write computed values. In these sections, mesh entities can be a node, an element face, or an element.

11.1.1 Subroutine Interface

The following interface is used for all user subroutines:

```
subroutine sub_name(int  num_objects,
                    int  num_values,
                    real  evaluation_time,
                    int  object_ids[],
                    real  output_values[],
                    int  output_flags[],
```

```
int error_code)
```

The name of the user subroutine, `sub_name`, is selected by the user. Avoid names for the subroutine that are longer than 10 characters. This may cause build problems on some systems.

A detailed description of the input and output parameters is provided in Table 11.1 and Table 11.2.

Table 11.1: Subroutine Input Parameters

Input Parameter	Data Type	Parameter Description
<code>num_objects</code>	Integer	Number of input mesh entities. For example, if the subroutine is a node set subroutine, this would be the number of nodes on which the subroutine will operate.
<code>num_values</code>	Integer	Number of return values. This is the number of values per mesh entity.
<code>evaluation_time</code>	Real	Time at which the subroutine should be evaluated. This may vary slightly from the current analysis time. For example, in explicit dynamics analyses, velocities are evaluated one-half time step ahead.
<code>object_ids</code> (<code>num_objects</code>)	Integer	Array of mesh-entity identification numbers. The array has a length of <code>num_objects</code> . The input numbers are the global numbers of the input objects. The object identification numbers can be used to query information about a mesh entity.

11.1.2 Query Functions

Sierra/SM follows a design philosophy for user subroutines that a minimal amount of information should be passed through the call list. Additional information may be queried from within the subroutine. A user subroutine may query a wide variety of information from Sierra/SM.

11.1.2.1 Parameter Query

A number of user subroutine parameters may be set as described in Section 11.2.2.3. These subroutine parameters can be obtained from the Sierra/SM input file via the query functions listed below.

```
aupst_get_real_param(string var_name, real var_value,
                    int error_code)
```

Table 11.2: Subroutine Output Parameters

Output Parameter	Data Type	Parameter Description
output_values (num_values, num_objects)	Integer	Array of output values computed by the subroutine. The number of output values will be either the number of mesh entities or some multiple of the number of mesh entities. For example, if there were six nodes (num_objects equals 6) and one value was to be computed per node, the length of output_values would be 6. Similarly, if there were six nodes (num_objects equals 6) and three values were to be computed for each node (as for acceleration, which has X-, Y-, and Z-components), the length of output_values would be 18.
output_flags (num_objects)	Integer	Array of returned flags for each set of data values. When used, this array will generally have a length of num_objects. The usage of the flags depends on subroutine type; the flags are currently used only for element death and for kinematic boundary conditions. For the kinematic boundary conditions (displacement, velocity, acceleration) a flag of -1 means ignore the constraint, a flag of 0 means set the absolute constraint value, and a flag of 1 means set the constraint with direction and distance.
error_code	Integer	Error code returned by the user subroutine. A value of 0 indicates no errors. Any value other than zero is an error. If the return value is nonzero, Sierra/SM will report the error code and terminate the analysis.

```

aupst_get_integer_param(string var_name, int var_value,
                        int error_code)

aupst_get_string_param(string var_name, string var_value,
                       int error_code)

```

All three of these subroutine calls are tied to a corresponding “parameter” command line that will appear in the Sierra/SM input file. The parameter command lines are described in [Section 11.2.2.3](#). These command lines are named based on the type of value they store, i.e., SUBROUTINE REAL PARAMETER, SUBROUTINE INTEGER PARAMETER, and SUBROUTINE STRING PARAMETER.

We will use the example of a real parameter to show how the subroutine call works in conjunction with the SUBROUTINE REAL PARAMETER command line. Suppose we have a real parameter radius that is set to a value of 2.75 on the SUBROUTINE REAL PARAMETER command line:

```
SUBROUTINE REAL PARAMETER: radius = 2.75
```

Also suppose we have a call to `aupst_get_real_parameter` in the user subroutine:

```
call aupst_get_real_parameter("radius", cyl_radius, error_code)
```

In the call to `aupst_get_real_parameter`, we have `var_name` set to `radius` and `var_value` defined as the real FORTRAN variable `cyl_radius`. The call to `aupst_get_real_parameter` will assign the value 2.75 to the FORTRAN variable `cyl_radius`. A similar pattern is followed for integer and string parameters.

The arguments for the parameter-related query functions are described in [Table 11.3](#), [Table 11.4](#), and [Table 11.5](#). The function is repeated before each table for easy reference.

```

aupst_get_real_param(string var_name, real var_value,
                    int error_code)

```

Table 11.3: aupst_get_real_param Arguments

Parameter	Usage	Data Type	Description
var_name	Input	String	Name of a real-valued subroutine parameter, as defined in the Sierra/SM input file via the SUBROUTINE REAL PARAMETER command line.
var_value	Output	Real	Name of a real variable to be used in the FORTRAN subroutine. The FORTRAN variable var_value will be set to the value specified by the SUBROUTINE REAL PARAMETER command line.
error_code	Output	Integer	Error code indicating status of retrieving the parameter value from the input file. If the retrieval is successful, error_code is set to 0. If the parameter is not found or is the wrong type, error_code is set to a value other than 0.

```

aupst_get_integer_param(string var_name, int var_value,
                        int error_code)

```

Table 11.4: aupst_get_integer_param Arguments

Parameter	Usage	Data Type	Description
var_name	Input	String	Name of an integer-valued subroutine parameter, as defined in the Sierra/SM input file via the SUBROUTINE INTEGER PARAMETER command line.
var_value	Output	Integer	Name of an integer variable to be used in the FORTRAN subroutine. The FORTRAN variable var_value will be set to the value specified by the SUBROUTINE INTEGER PARAMETER command line.
error_code	Output	Integer	Error code indicating status of retrieving the parameter value from the input file. If the retrieval is successful, error_code is set to 0. If the parameter is not found or is the wrong type, error_code is set to a value other than 0.


```

aupst_get_string_param(string var_name, string var_value,
                      int error_code)

```

Table 11.5: aupst_get_string_param Arguments

Parameter	Usage	Data Type	Description
var_name	Input	String	Name of a string-valued subroutine parameter, as defined in the Sierra/SM input file via the SUBROUTINE STRING PARAMETER command line.
var_value	Output	String	Name of a string variable to be used in the FORTRAN subroutine. The FORTRAN variable var_value will be set to the value specified by the SUBROUTINE STRING PARAMETER command line.
error_code	Output	Integer	Error code indicating status of retrieving the parameter value from the input file. If the retrieval is successful, error_code is set to 0. If the parameter is not found or is the wrong type, error_code is set to a value other than 0.

11.1.2.2 Function Data Query

The function data query routine listed below may be used for extracting data from a function that is defined in a `FUNCTION` command block in the Sierra/SM input file. This query allows the user to directly access information stored in a function defined in the Sierra/SM input file.

```

aupst_evaluate_function(string func_name, real input_times[],
                      int num_times, real output_data[])

```

The arguments for this function are described in Table 11.6.

11.1.2.3 Time Query

The time query function can be used to determine the current analysis time. This is the time associated with the new time step. This time may not be equivalent to the `evaluation_time` argument passed into the subroutine (see Section 11.1.1, Table 11.1) as some boundary conditions need to be evaluated at different times than others. The parameter of the time query function listed below is given in Table 11.7.

```

aupst_get_time(real time)

```

Table 11.6: aupst_evaluate_function Arguments

Parameter	Usage	Data Type	Description
func_name	Input	String	Name of the function to look up.
input_times (num_times)	Input	Real	Array of times used to extract values of the function.
num_times	Input	Integer	Length of the array input_times.
output_data (num_times)	Output	Real	Array of output values of the named function at the specified times.

Table 11.7: aupst_get_time Argument

Parameter	Usage	Data Type	Description
time	Output	Real	Current analysis time.

11.1.2.4 Field Variables

Field variables (displacements, stresses, etc.) may be defined on groups of mesh entities. Many queries are available for getting and putting field variables. These queries involve passing in a set of mesh-entity identification numbers to receive field values on the mesh entities. There are query functions to check for the existence and size of a field, functions to retrieve the field values, and functions to store new variables in a field. The field query functions listed below can be used to extract any nodal or element variable field.

```

aupst_check_node_var(int num_nodes, int num_components,
                    int node_ids[], string var_name,
                    int error_code)

aupst_check_elem_var(int num_elems, int num_components,
                    int elem_ids[], string var_name,
                    int error_code)

aupst_get_node_var(int num_nodes, int num_components,
                  int node_ids[], real return_data[],
                  string var_name, int error_code)

aupst_get_elem_var(int num_elems, int num_components,
                  int elem_ids[], real return_data[],
                  string var_name, int error_code)

aupst_get_elem_var_offset(int num_elems, int num_components,
                         int offset, int elem_ids[],
                         real return_data[], string var_name,
                         int error_code)

```

```

aupst_put_node_var(int num_nodes, int num_components,
                  int node_ids[], real new_data[],
                  string var_name, int error_code)

aupst_put_elem_var(int num_elems, int num_components,
                  int elem_ids[], real new_data[],
                  string var_name, int error_code)

aupst_put_elem_var_offset(int num_elems, int num_components,
                          int offset, int elem_ids[],
                          real new_data[], string var_name,
                          int error_code)

```

The arrays where data are stored are static arrays. These arrays of a set size will be declared at the beginning of a user subroutine. The query functions to check for the existence and size of a field can be used to ensure that the size of the array of information being returned from Sierra/SM does not exceed the size of the array allocated by the user.

The arguments to field query functions are defined in Table 11.8 through Table 11.15. The function is repeated before each table for easy reference.

The `var_name` strings may be the name of any variable that could be used for code output and additionally may include the component syntax described in 9.1.

```

aupst_check_node_var(int num_nodes, int num_components,
                    int node_ids[], string var_name,
                    int error_code)

```

Table 11.8: aupst_check_node_var Arguments

Parameter	Usage	Data Type	Description
num_nodes	Input	Integer	Number of nodes used to extract field information.
num_components	Output	Integer	Number of components in the field information. A displacement field at a node has three components, for example.
node_ids (num_nodes)	Input	Integer	Array of size num_nodes listing the node identification number for each node where field information will be retrieved.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.

```

aupst_check_elem_var(int num_elems, int num_components,
                    int elem_ids[], string var_name,
                    int error_code)

```

Table 11.9: aupst_check_elem_var Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements used to extract field information.
num_components	Output	Integer	Number of components in the field information. A stress field for a an eight-node hexahedron element has six components, for example.
elem_ids (num_elems)	Input	Integer	Array of size num_elems listing the element identification number for each element where field information will be retrieved.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.

```

aupst_get_node_var(int num_nodes, int num_components,
                  int node_ids[], real return_data[],
                  string var_name, int error_code)

```

Table 11.10: aupst_get_node_var Arguments

Parameter	Usage	Data Type	Description
num_nodes	Input	Integer	Number of nodes used to extract field information.
num_components	Input	Integer	Number of components in the field information. A displacement field at a node has three components, for example.
node_ids (num_nodes)	Input	Integer	Array of size num_nodes listing the node identification number for each node where field information will be retrieved.
return_data (num_components, num_nodes)	Output	Real	Array of size num_components $\times$ num_nodes containing the field data at each node.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.

```

aupst_get_elem_var(int num_elems, int num_components,
                  int elem_ids[], real return_data[],
                  string var_name, int error_code)

```

Table 11.11: aupst_get_elem_var Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements used to extract field information.
num_components	Input	Integer	Number of components in the field information. A stress field for a an eight-node hexahedron element has six components, for example.
elem_ids (num_elems)	Input	Integer	Array of size num_elems listing the element identification number for each element where field information will be retrieved.
return_data (num_components, num_elems)	Output	Real	Array of size num_components $\times$ num_elems containing the field data for each element.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.

```

aupst_get_elem_var_offset(int num_elems, int num_components,
                          int offset, int elem_ids[],
                          real return_data[], string var_name,
                          int error_code)

```

Table 11.12: aupst_get_elem_var_offset Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements used to extract field information.
num_components	Input	Integer	Number of components in the field information. A stress field for an eight-node hexahedron element has six components, for example.
offset	Input	Integer	Offset into var_name field variable at which to get data.
elem_ids (num_elems)	Input	Integer	Array of size num_elems listing the element identification number for each element where field information will be retrieved.
return_data (num_components, num_elems)	Output	Real	Array of size num_components × num_elems containing the field data for each element.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.


```

aupst_put_node_var(int num_nodes, int num_components,
                  int node_ids[], real new_data[],
                  string var_name, int error_code)

```

Table 11.13: aupst_put_node_var Arguments

Parameter	Usage	Data Type	Description
num_nodes	Input	Integer	Number of nodes for which the user will specify the field data.
num_components	Input	Integer	Number of components in the field information. A displacement field at a node has three components, for example.
node_ids (num_nodes)	Input	Integer	Array of size num_nodes listing the node identification number for each node where field information will be retrieved.
new_data (num_components, num_nodes)	Input	Real	Array of size num_components $\times$ num_nodes containing the new data for the field.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.

```

aupst_put_elem_var(int num_elems, int num_components,
                  int elem_ids[], real new_data[],
                  string var_name, int error_code)

```

Table 11.14: aupst_put_elem_var Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements for which the user will specify the field data.
num_components	Input	Integer	Number of components in the field information. A stress field for a an eight-node hexahedron element has six components, for example.
elem_ids (num_elems)	Input	Integer	Array of size num_elems listing the element identification number for each element where field information will be retrieved.
new_data (num_components, num_elems)	Input	Real	Array of size num_components $\times$ num_elems containing the new data for the field.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.

```

aupst_put_elem_var_offset(int num_elems, int num_components,
                          int offset, int elem_ids[],
                          real new_data[], string var_name,
                          int error_code)

```

Table 11.15: aupst_put_elem_var_offset Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements for which the user will specify the field data.
num_components	Input	Integer	Number of components in the field information. A stress field for an eight-node hexahedron element has six components, for example.
offset	Input	Integer	Offset into var_name field variable at which to put data.
elem_ids (num_elems)	Input	Integer	Array of size num_elems listing the element identification number for each element where field information will be retrieved.
new_data (num_components, num_elems)	Input	Real	Array of size num_components × num_elems containing the new data for the field.
var_name	Input	String	Name of the field variable. The field variable must be a defined Sierra/SM variable.
error_code	Output	Integer	Error code indicating status of retrieving the field. If the retrieval is successful, error_code is set to 0. If a nonzero value is returned for error_code, the field variable does not exist or is not defined on one or more of the input nodes.

11.1.2.5 Global Variables

Global variables may be extracted or set from user subroutines. A global variable has a single value for a given region. Global variables have limited support for parallel operations. The subroutine `aupst_put_global_var` attempts to immediately modify and optionally perform a parallel reduction on the value of a global variable. `aupst_put_global_var` can specify the variable reduction type “sum”, “min”, “max”, and “none”. If the “none” option is passed to the routine no global reduction operation will take place, only the processor local copy of the global variable will be modified. If any option other than “none” is used care must be taken to call

`aupst_put_global_var` on all processors at the same time with the same arguments. Failure to call the routine from all processors will result in the code hanging. For some types of subroutines this is not possible or reliable. For example, a boundary condition subroutine may not be called at all on a processor that contains no nodes in the set of nodes assigned to the boundary condition. It is recommended that the `aupst_put_global_var` function only be used in conjunction with a user subroutine referenced in a `USER OUTPUT` command block (Section 9.3.2).

Only user-defined global variables may be modified by the user subroutine (see Section 11.2.4). However, any global variable that exists on the region may be checked or extracted. The following subroutine calls pertain to global variables:

```
aupst_check_global_var(int num_comp, string var_name
                      int error_code)

aupst_get_global_var(int num_comp, real return_data,
                    string var_name, int error_code)

aupst_put_global_var(int num_comp, real input_data,
                    string var_name,
                    string reduction_type,
                    int error_code)
```

The arguments for subroutine calls pertaining to global variables are defined in Table 11.16 through Table 11.18. The call is repeated before each table for easy reference.

```
aupst_check_global_var(int num_comp, string var_name
                      int error_code)
```

Table 11.16: aupst_check_global_var Arguments

Parameter	Usage	Data Type	Description
num_comp	Output	Integer	Number of components of the global variable.
var_name	Input	String	Name of the global variable.
error_code	Output	Integer	Error code indicating status of accessing the global variable. If there is no error in accessing this variable, error_code is set to 0. A nonzero value of error_code means the global variable does not exist or in some way cannot be accessed.

```
aupst_get_global_var(int num_comp, real return_data,
                    string var_name, int error_code)
```

Table 11.17: aupst_get_global_var Arguments

Parameter	Usage	Data Type	Description
num_comp	Input	Integer	Number of components of the global variable.
return_data	Output	Real	Value of the global variable.
var_name	Input	String	Name of the global variable.
error_code	Output	Integer	Error code indicating status of accessing the global variable. If there is no error in accessing this variable, error_code is set to 0. A nonzero value of error_code means the global variable does not exist or in some way cannot be accessed.

```

aupst_put_global_var(int num_comp, real input_data,
                    string var_name,
                    string reduction_type,
                    int error_code)

```

Table 11.18: aupst_put_global_var Arguments

Parameter	Usage	Data Type	Description
num_comp	Input	Integer	Number of components of the global variable.
input_data	Input	Real	New value of the global variable.
var_name	Input	String	Name of the global variable.
reduction_type	Input	String	Type of parallel reduction to perform on the variable. Options are “sum”, “min”, “max”, and “none”.
error_code	Output	Integer	Error code indicating status of accessing the global variable. If there is no error in accessing this variable, error_code is set to 0. A nonzero value of error_code means the global variable does not exist, in some way cannot be accessed, or may not be overwritten.

11.1.2.6 Topology Extraction

The element and surface subroutines operate on groups of elements or faces. The elements and faces may have a variety of topologies. Topology queries can be used to get topological data about elements and faces. The topology of an object is represented by an integer. The integer is formed from a function of the number of dimensions, vertices, and nodes of an object. The topology of an object is given by:

```

topology = num_node + 100 * num_vert + 10000 * num_dim

```

In a FORTRAN routine, the number of nodes can easily be extracted with the mod function:

```

num_node = mod(topo, 100)
num_vert = mod(topo / 100, 100)
num_dim  = mod(topo / 10000, 100)

```

Table 11.19 lists the topologies currently in use by Sierra/SM.

The following topology query functions are available in Sierra/SM:

```

aupst_get_elem_topology(int num_elems, int elem_ids[],

```

Table 11.19: Topologies Used by Sierra/SM

Topology	Element / Face Type
00101	One-node particle
10202	Two-node beam, truss, or damper
20404	Four-node quadrilateral
20303	Three-node triangle
20304	Four-node triangle
20306	Six-node triangle
30404	Four-node tetrahedron
30410	Ten-node tetrahedron
30808	Eight-node hexahedron
30606	Six-node wedge
30505	Five-node pyramid

```

                                int topology[], int error_code)

aupst_get_elem_nodes(int num_elems, int elem_ids[],
                    int elem_node_ids[], int error_code)

aupst_get_face_topology(int num_faces, int face_ids[],
                       int topology[], int error_code)

aupst_get_face_nodes(int num_faces, int face_ids[],
                    int face_node_ids[], int error_code)

```

The arguments for the topology extraction functions are defined in Table 11.20 through Table 11.23. The function is repeated before each table for easy reference.

```
aupst_get_elem_topology(int num_elems, int elem_ids[],
                        int topology[], int error_code)
```

Table 11.20: aupst_get_elem_topology Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements from which the topology will be extracted.
elem_ids (num_elems)	Input	Integer	Array of length num_elems listing the element identification for each element from which the topology will be extracted.
topology (num_elems)	Output	Integer	Array of length num_elems that has the topology for each element. See Table 8.18.
error_code	Output	Integer	Error code indicating status of retrieving the element identification numbers. If the retrieval is successful, error_code is set to 0. A nonzero value is returned for error_code if one of the element identification numbers is not valid.


```

aupst_get_elem_nodes(int num_elems, int elem_ids[],
                    int elem_node_ids[], int error_code)

```

Table 11.21: aupst_get_elem_nodes Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements from which the topology will be extracted.
elem_ids (num_elems)	Input	Integer	Array of length num_elems listing the element identification for each element from which the topology will be extracted.
elem_node_ids (number of nodes for element type $\times$ num_elems)	Output	Integer	Array containing the node identification numbers for each element requested. The length of the array is the total number of nodes contained in all elements. If the elements are eight-node hexahedra, then the number of nodes will be $8 \times \text{num\_elems}$. The first set of eight entries in the array will be the eight nodes defining the first element. The second set of eight entries will be the eight nodes defining the second element, and so on.
error_code	Output	Integer	Error code indicating status of retrieving the element identification numbers. If the retrieval is successful, error_code is set to 0. A nonzero value is returned for error_code if one of the element identification numbers is not valid.

```

aupst_get_face_topology(int num_faces, int face_ids[],
                        int topology[], int error_code)

```

Table 11.22: aupst_get_face_topology Arguments

Parameter	Usage	Data Type	Description
num_faces	Input	Integer	Number of faces from which the topology will be extracted.
face_ids (num_faces)	Input	Integer	Array of length num_faces listing the face identification for each face from which the topology will be extracted.
topology (num_faces)	Output	Integer	Array of length num_faces containing the output topologies of each face.
error_code	Output	Integer	Error code indicating status of retrieving the face identification numbers. If the retrieval is successful, error_code is set to 0. A nonzero value is returned for error_code if one of the face identification numbers is not valid.

```

aupst_get_face_nodes(int num_faces, int face_ids[],
                    int face_node_ids[], int error_code)

```

Table 11.23: aupst_get_face_nodes Arguments

Parameter	Usage	Data Type	Description
num_faces	Input	Integer	Number of faces from which the topology will be extracted.
face_ids (num_faces)	Input	Integer	Array of length num_faces listing the face identification for each face from which the topology will be extracted.
face_node_ids (number of nodes for face type $\times$ num_faces)	Output	Integer	Array containing the node identification numbers for each face requested. The length of the array is the total number of nodes contained in all faces. If the faces are four-node quadrilaterals, then the number of nodes will be $4 \times \text{num\_faces}$. The first set of four entries in the array will be the four nodes defining the first face. The second set of four entries will be the four nodes defining the second face, and so on.
error_code	Output	Integer	Error code indicating status of retrieving the face identification numbers. If the retrieval is successful, error_code is set to 0. A nonzero value is returned for error_code if one of the face identification numbers is not valid.

11.1.3 Miscellaneous Query Functions

Many miscellaneous query functions are available for computing some commonly used quantities.

```

aupst_get_elem_centroid(int num_elems, int elem_ids[],
                      real centroids, int error_code)

```

```

aupst_get_point(string point_name, real point_coords,
               int error_code)

```

```

aupst_get_proc_num(proc_num)

```

The arguments for the miscellaneous query functions are defined in Table 11.24 through Table 11.26. The function is repeated before each table for easy reference.

```
aupst_get_elem_centroid(int num_elems, int elem_ids[],
                        real centroids[], int error_code)
```

Table 11.24: aupst_get_elem_centroid Arguments

Parameter	Usage	Data Type	Description
num_elems	Input	Integer	Number of elements for which to extract the topology.
elem_ids (num_elems)	Input	Integer	Array of length num_elems listing the element identification for each element for which the centroid will be computed.
centroids (3, num_elems)	Output	Real	Array of length $3 \times \text{num\_elems}$ containing the centroid of each element.
error_code	Output	Integer	Error code indicating status of retrieving the element identification numbers. If the retrieval is successful, error_code is set to 0. A nonzero value is returned for error_code if one of the element identification numbers is not valid.

```
aupst_get_point(string point_name, real point_coords,
                int error_code)
```

Table 11.25: aupst_get_point Arguments

Parameter	Usage	Data Type	Description
point_name	Input	String	SIERRA name for a given point.
point_coords (3)	Output	Real	Array of length 3 containing the <i>x</i> , <i>y</i> , and <i>z</i> coordinates of the point.
error_code	Output	Integer	Error code indicating status of retrieving the point. If the retrieval is successful, error_code is set to 0. A nonzero value is returned for error_code if the point cannot be found

```
aupst_get_proc_num(proc_num)
```

Table 11.26: aupst_get_proc_num Arguments

Parameter	Usage	Data Type	Description
proc_num	Output	Integer	Processor number of the calling process. This number can be used for informational purposes. A common example is that output could only be written by a single processor, e.g., processor 0, rather than by all processors.

11.2 Command File

In addition to the actual user subroutine, you will need to add command lines to your input file to make use of your user subroutine. This section describes the command lines that are used in conjunction with user subroutines. This section also describes two additional command blocks, `TIME STEP INITIALIZATION` and `USER VARIABLE`. The `TIME STEP INITIALIZATION` command block lets you execute a user subroutine at the beginning of a time step as opposed to some later time. The `USER VARIABLE` command block can be used in conjunction with user subroutines or for user-defined output.

11.2.1 Subroutine Identification

As described in Section 2.1.4, there is one command line associated with the user subroutine functionality that must be provided in the SIERRA scope:

```
USER SUBROUTINE FILE = <string>file_name
```

The named file may contain one or more user subroutines. The file must have an extension of “.F”, as in `blast.F`.

11.2.2 User Subroutine Command Lines

```
{begin command block}
  NODE SET SUBROUTINE = <string>subroutine_name |
    SURFACE SUBROUTINE = <string>subroutine_name |
    ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
  SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
  SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
  SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
  SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
{end command block}
```

A number of user subroutine command lines will appear in some Sierra/SM command block. User subroutine commands can appear in boundary condition, element death (only for explicit dynamics), user output, and state initialization command blocks. The possible command lines are shown above. The following sections describe the command lines related to user subroutines.

11.2.2.1 Type

User subroutines are currently available in three general types: node set, surface, and element.

Node set subroutines operate on groups of nodes. The command line for defining a node set subroutine is:

```
NODE SET SUBROUTINE = <string>subroutine_name
```

where `subroutine_name` is the name of the user subroutine. The name is case sensitive. A node

set subroutine will operate on all nodes contained in an associated mechanics instance.

Surface subroutines work on groups of surfaces. A surface may be an external face of a solid element or the face of a shell element associated with either the positive or negative normal for the surface of the shell. The command line for defining a surface subroutine is:

```
SURFACE SUBROUTINE = <string>subroutine_name
```

Element block subroutines work on groups of elements. The command line for defining an element block subroutine is:

```
ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
```

An element may be a solid element such as a hexahedron or a two-dimensional element such as a shell.

Different Sierra/SM features may accept one or more types of user subroutines. Only one subroutine is allowed per command block.

11.2.2.2 Debugging

Subroutines may be run in a special debugging mode to help catch memory errors. For example, there is a potential for a user subroutine to write outside of its allotted data space by writing beyond the bounds of an input or output array. Generally, this causes Sierra/SM to crash, but it also has the potential to introduce other hard-to-trace bugs into the Sierra/SM analysis. Subroutines run in debug mode require more memory and more processing time than subroutines not run in debug mode.

Subroutine debugging is on by default in debug executables. It can be turned off with the following command line:

```
SUBROUTINE DEBUGGING OFF
```

Subroutine debugging is off by default in optimized executables. It can be turned on with the following command line:

```
SUBROUTINE DEBUGGING ON
```

11.2.2.3 Parameters

All user subroutines have the ability to use parameters. Parameters are defined in the input file and are quickly accessible by the user subroutine during run time. Parameters are a way of making a single user subroutine much more versatile. For example, a user subroutine could be written to define a periodic loading on a structure. A parameter for the subroutine could be defined specifying the frequency of the function. In this way, the same subroutine can be used in different parts of the model, and the subroutine behavior can be modified without recompiling the program. These command lines are placed within the scope of the command block in which the user subroutine is specified.

Real-valued parameters can be stored with the following command line:

```
SUBROUTINE REAL PARAMETER: <string>param_name  
= <real>param_value
```

Integer-valued parameters can be stored with the following command line:

```
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
```

String-valued parameters can be stored with the following command line:

```
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

Any number of subroutine parameters may be defined. The subroutine parameters may be defined in any order within the command block. The user subroutine may request the values of the parameters but is not required to use them or even have any knowledge of their existence. An example of subroutine usage with parameters is as follows:

```
BEGIN PRESSURE
    SURFACE = surface_1
    SURFACE SUBROUTINE = blast_pressure
    SUBROUTINE REAL PARAMETER: blast_time = 1.2
    SUBROUTINE REAL PARAMETER: blast_power = 1.3e+07
    SUBROUTINE STRING PARAMETER: formulation = alpha
    SUBROUTINE INTEGER PARAMETER: decay_exponent = 2
    SUBROUTINE DEBUGGING ON
END PRESSURE
```

In the above example, four parameters are associated with the subroutine `blast_pressure`. Two of the parameters are real (`blast_time` and `blast_power`), one of the parameters is a string (`formulation`), and one of the parameters is an integer (`decay_exponent`). To access the parameters in the user subroutine, the user will need to include interface calls described in previous sections.

11.2.3 Time Step Initialization

```
BEGIN TIME STEP INITIALIZATION
    # mesh-entity set commands
    NODE SET = <string list>nodelist_names
    SURFACE = <string list>surface_names
    BLOCK = <string list>block_names
    INCLUDE ALL BLOCKS
    REMOVE NODE SET = <string list>nodelist_names
    REMOVE SURFACE = <string list> surface_names
    REMOVE BLOCK = <string list>block_names
    #
    # read mesh variable commands
    READ VARIABLE = <string> mesh_var_name
    TYPE = <string>NODE|ELEMENT
        [<string>REAL|INTEGER LENGTH = <integer>length] |
        [<string>SYM_TENSOR|FULL_TENSOR|VECTOR]
    #
```



```

# user subroutine commands
NODE SET SUBROUTINE = <string>subroutine_name |
  SURFACE SUBROUTINE = <string>sub_name |
  ELEMENT BLOCK SUBROUTINE = <string>sub_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
  = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
  = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
  = <string>param_value
#
# additional command
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names

END TIME STEP INITIALIZATION

```

The `TIME STEP INITIALIZATION` command block, which appears in the region scope, is used to control variables while an analysis is running. This can be done by reading a variable from a separate mesh file or by running a user subroutine. These variables can then be used to compute quantities used by other command types. For example, if the traction on a surface was dependent on the area, the time step initialization subroutine could be used to calculate the area, and that area could be stored and later read when calculating the traction. The user initialization subroutine will pass the specified mesh objects to the subroutine for use in calculating some value.

The `TIME STEP INITIALIZATION` command block contains two groups of commands—mesh entity set and user subroutine. In addition to the command lines in these command groups, there is an additional command line: `ACTIVE PERIODS` or `INACTIVE PERIODS`. Following are descriptions of the different command groups and the `ACTIVE PERIODS` or `INACTIVE PERIODS` command line.

11.2.3.1 Mesh-Entity Set Commands

The mesh-entity set commands portion of the `TIME STEP INITIALIZATION` command block specifies the nodes, element faces, or elements associated with the particular subroutine run at the beginning of the applicable time steps. This portion of the command block can include some combination of the following command lines:

```

NODE SET = <string list>nodelist_names
SURFACE = <string list>surface_names
BLOCK = <string list>block_names
INCLUDE ALL BLOCKS
REMOVE NODE SET = <string list>nodelist_names
REMOVE SURFACE = <string list> surface_names
REMOVE BLOCK = <string list>block_names

```

These command lines, taken collectively, constitute a set of Boolean operators for constructing a set of nodes, element faces, or elements. See Section 6.1 for more information about the use of these command lines for mesh entities. There must be at least one `NODE SET`, `SURFACE`, `BLOCK`, or `INCLUDE ALL BLOCKS` command line in the command block.

11.2.3.2 Read Mesh Variable Commands

The following command lines are related to copying a mesh variable from another exodus file to the current exodus mesh.

```
READ VARIABLE = <string> mesh_var_name
TYPE = <string>NODE|ELEMENT
      [<string>REAL|INTEGER LENGTH = <integer>length] |
      [<string>SYM_TENSOR|FULL_TENSOR|VECTOR]
```

The `READ VARIABLE` command is used to specify the name of the variable to be read from the other exodus mesh. The file the variable will be read from is specified in the `DATABASE NAME` command within the `FINITE ELEMENT MODEL` block. The `TYPE` command is used to specify the type of data the field is associated with. Once these two commands are specified, the variable will show up on the new exodus mesh with the same name.

11.2.3.3 User Subroutine Commands

The following command lines are related to the user subroutine specification:

```
NODE SET SUBROUTINE = <string>subroutine_name |
SURFACE SUBROUTINE = <string>subroutine_name |
ELEMENT BLOCK SUBROUTINE = <string>subroutine_name
SUBROUTINE DEBUGGING OFF | SUBROUTINE DEBUGGING ON
SUBROUTINE REAL PARAMETER: <string>param_name
    = <real>param_value
SUBROUTINE INTEGER PARAMETER: <string>param_name
    = <integer>param_value
SUBROUTINE STRING PARAMETER: <string>param_name
    = <string>param_value
```

Only one of the first three command lines listed above can be specified in the command block. The particular command line selected depends on the mesh-entity type of the variable being initialized. For example, variables associated with nodes would be initialized if you are using the `NODE SET SUBROUTINE` command line, variables associated with faces if you are using the `SURFACE SUBROUTINE` command line, and variables associated with elements if you are using the `ELEMENT BLOCK SUBROUTINE` command line. The string `subroutine_name` is the name of a FORTRAN subroutine that is written by the user.

Following the selected subroutine command line are other command lines that may be used to implement the user subroutine option. These command lines are described in Section 11.2.2 and

consist of SUBROUTINE DEBUGGING OFF, SUBROUTINE DEBUGGING ON, SUBROUTINE REAL PARAMETER, SUBROUTINE INTEGER PARAMETER, and SUBROUTINE STRING PARAMETER. Examples of using these command lines are provided throughout Chapter 11.

11.2.3.4 Additional Command

The ACTIVE PERIODS or INACTIVE PERIODS command line can optionally appear in the TIME STEP INITIALIZATION command block:

```
ACTIVE PERIODS = <string list>period_names
INACTIVE PERIODS = <string list>period_names
```

The ACTIVE PERIODS or INACTIVE PERIODS command line is used to activate or deactivate the running of the user subroutine at the beginning of every time step for certain time periods. See Section 2.6 for more information about this optional command line.

11.2.4 User Variables

```
BEGIN USER VARIABLE <string>var_name
  TYPE = <string>NODE|ELEMENT|GLOBAL
        [<string>REAL|INTEGER LENGTH = <integer>length] |
        [<string>SYM_TENSOR|FULL_TENSOR|VECTOR]
  GLOBAL OPERATOR = <string>SUM|MIN|MAX
  INITIAL VALUE = <real list>values
  INITIAL VARIATION = <real list>values LINEAR DISTRIBUTION
  USE WITH RESTART
END [USER VARIABLE <string>var_name]
```

The USER VARIABLE command block is used to create a user-defined variable. This kind of variable may be used for scratch space in a user subroutine or for some user-defined output. A user-defined variable may be output to the results file or the history file just like any other defined variable; i.e., a user-defined variable once defined by the USER VARIABLE command block can be specified in a USER OUTPUT command block, a RESULTS OUTPUT command block, and a HISTORY OUTPUT command block.

User-defined variables are associated with mesh entities. For example, a node variable will exist at every node of the model. An element variable will exist on every element of the model. A global variable will have a single value for the entire model.

If the user-defined variable functionality is used in conjunction with restart, the USE WITH RESTART command line must be included.

The USER VARIABLE command block is placed within a Sierra/SM region. The command block begins with the input line:

```
BEGIN USER VARIABLE <string>var_name
```

and ends with the input line:

```
END [USER VARIABLE <string>var_name]
```

where var_name is a user-selected name for the variable.

In the above command block:

- A user-defined variable has an associated type that is specified by the `TYPE` command line, which itself contains several parameters. The `TYPE` command line is required.
 1. The variable must be a nodal quantity, an element quantity, or a global quantity. The options `NODE`, `ELEMENT`, and `GLOBAL` determine whether the variable will be a nodal, element, or global quantity. One of these options must appear on the `TYPE` command line.
 2. The user-defined variable can be either an integer or a real, as specified by the `INTEGER` or `REAL` option.
 3. The length of the variable must be set by using one of the options `SYM_TENSOR`, `FULL_TENSOR`, `VECTOR`, or `LENGTH = <integer>length`. If the `LENGTH` option is used, the user must specify whether the variable is an integer number or a real number by using the `INTEGER` or `REAL` option. If the `SYM_TENSOR` option is used, the variable has six real components. If the `FULL_TENSOR` is used, the variable has nine real components. If the `VECTOR` option is used, the variable has three real components. The three options `SYM_TENSOR`, `FULL_TENSOR`, and `VECTOR` all imply real numbers, and thus the `REAL` option need not be included in the command line when one of these three options is specified.

Some examples of the `TYPE` command line follow:

```
type = global real length = 1
type = element tensor
type = element real length = 3
type = node sym_tensor
type = node vector
```

- If you use the `GLOBAL` option on the `TYPE` command line, a global variable is created, and this global variable must be given an associated reduction type, which is specified by the `GLOBAL OPERATOR` command line. The reduction type tells Sierra/SM how to reduce the individual values stored on each processor to a mesh global value. Global reductions are performed at the end of each time step. The `SUM` operator sums all processor variable contributions. The `MAX` operator takes the maximum value among all processor variable contributions. The `MIN` operator takes the minimum value among all processor variable contributions.
- One or more initial values may be specified for the user-defined variable in the `INITIAL VALUE` command line. The number of initial values specified should be the same as the length of the variable, as specified in the `TYPE` command line either explicitly via the `LENGTH` option or implicitly via the `SYM_TENSOR`, `FULL_TENSOR`, or `VECTOR` option. The initial values will be copied to the variable space on every mesh object on which the variable is defined. Only real type variables may be given initial values at this time.
- The initial value of the user variable can be given some random distribution by use of the `INITIAL VARIATION` command line. If the `INITIAL VARIATION` command is used the `INITIAL VALUE` command must also be used. In addition the number of values specified in the initial variation command must exactly match the number of values specified in the initial value command. When the initial variation command is used the initial values of the variable will be set as initial value plus or minus some random factor times the initial

variation. Currently the only random distribution supported is the linear distribution. With linear distribution the random values will be distributed evenly from initial value minus variation to initial value plus variation.

- All intrinsic type options such as `REAL`, `INTEGER`, `SYM_TENSOR`, `FULL_TENSOR`, `VECTOR` and the `LENGTH` option can be used with any of the mesh entity options (`NODE`, `ELEMENT`, `GLOBAL`).
- As indicated previously, if the user-defined variable functionality is used in conjunction with restart, the `USE WITH RESTART` command line must be included.

11.3 Compilation and Execution

Running a code with user subroutines is a two-step process. First, an executable version of Sierra/SM that recognizes the user subroutines must be created. This executable must then be used for a Sierra/SM analysis with an input file that incorporates the proper user subroutine command lines.

How the above two steps are carried out is site- and installation-specific, but an example demonstrating this process on various systems at Sandia using SIERRA—a general code framework and code management system at Sandia—command lines is provided.

For the first step, you will need the user subroutine, in a FORTRAN file, and a Sierra/SM input file that makes use of the user subroutine. You will use a system command line of the general form shown below.

```
% sierra --make adagio -i <string>input_file_name
```

Suppose that you have a subdirectory in your area called `test` and you wish to incorporate a user subroutine called `blast_load`. The actual user subroutine will be in a file called `blast_load.F`, and the associated input file will be called `blast_load_1.i`. Both of these files will be in the directory `test`. In the input file, you will have the following command line in the SIERRA scope:

```
USER SUBROUTINE FILE = blast_load.F
```

You will also have some subset of the command lines described in the previous section in your Sierra/SM input file. The specific form of the system command line to execute the first step of the user subroutine process is shown below.

```
% sierra --make adagio -i blast_load_1.i
```

The above command will create a local version of Sierra/SM in a local directory named `UserSubsProject`. The system command line to run the local version of Presto is shown below.

```
% sierra ./UserSubsProject/bin/adagio -i <string>input_file_name
```

The specific form of the system command line you will execute in the subdirectory `test` is shown below.

```
% sierra ./UserSubsProject/bin/adagio -i blast_load_1.i
```

The second command line runs Sierra/SM using `blast_load_1.i` as an input file and utilizes the user subroutines in the process. Again, all of this is a site-specific example. You must determine how Sierra/SM is set up at your installation to determine what system command lines are necessary to build Sierra/SM with user subroutines and then use this version of Sierra/SM.

11.4 Examples

11.4.1 Pressure as a Function of Space and Time

(The following example provides functionality—a blast load on a surface—more applicable to Presto than Adagio. It is included in both manuals as it is instructive in the general use of a user-defined subroutine.)

The following code is an example of a user subroutine to compute blast pressures on a group of faces. The blast pressure simulates a blast occurring at a specified position and time. The blast wave radiates out from the center of the blast and dissipates over time. This subroutine is included in the input file as follows:

```
#In the SIERRA scope:
user subroutine file = blast_load.F

#In the region scope:
begin pressure
  surface = surface_1
  surface subroutine = blast_load
  subroutine real parameter: pos_x = 5.0
  subroutine real parameter: pos_y = 5.0
  subroutine real parameter: pos_z = 1.6
  subroutine real parameter: wave_speed = 1.5e+02
  subroutine real parameter: blast_time = 0.0
  subroutine real parameter: blast_energy = 1.0e+09
  subroutine real parameter: blast_wave_width = 0.75
end pressure
```

The FORTRAN 77 subroutine listing follows. It would be possible to increase the speed of this subroutine by calling the topology functions (see [Section 11.1.2.6](#)) on groups of elements, though this would increase subroutine complexity.

```
c
c Subroutine to simulate a blast load on a surface
c
      subroutine blast_load(num_faces, num_vals,
&   eval_time, faceID, pressure, flags, err_code)

      implicit none
c
c Subroutine input arguments
c
      integer num_faces
      double precision eval_time
      integer faceID(num_faces)
      integer num_vals
```

```

c
c Subroutine output arguments
c
      double precision pressure(num_vals, num_faces)
      integer flags(num_faces)
      integer err_code
c
c Variables to hold the subroutine parameters
c
      double precision pos_x, pos_y, pos_z, wave_speed,
&
      blast_time, blast_energy,
&
      blast_wave_width
c
c Local variables
c
      integer iface, inode
      integer cur_face_id, face_topo, num_nodes
      integer num_comp_check
      double precision dist, blast_o_rad, blast_i_rad
      double precision blast_volume, blast_pressure
      integer query_error
      double precision face_center(3)
c
c Create some static variables to hold queried
c information. Assume no face has more than 10
c nodes
c
      double precision face_nodes(10)
      double precision face_coords(3, 10)
c
c Extract the subroutine parameters
c
      call aupst_get_real_param("pos_x",pos_x,query_error)
      call aupst_get_real_param("pos_y",pos_y,query_error)
      call aupst_get_real_param("pos_z",pos_z,query_error)
      call aupst_get_real_param("wave_speed",wave_speed,
&
      query_error)
      call aupst_get_real_param("blast_energy",
&
      blast_energy,query_error)
      call aupst_get_real_param("blast_time",
&
      blast_time,query_error)
      call aupst_get_real_param("blast_wave_width",
&
      blast_wave_width, query_error)
c
c Determine the outer radius of the blast wave
c

```



```

        blast_o_rad = (eval_time - blast_time) * wave_speed
        if(blast_o_rad .le. 0.0) return;
c
c Determine the inner radius of the blast wave
c
        blast_i_rad = blast_o_rad - blast_wave_width
        if(blast_i_rad .le. 0.0) blast_i_rad = 0.0
c
c Determine the total volume the blast wave occupies
c
        blast_volume = 3.1415 * (4.0/3.0) *
&                (blast_o_rad**2 - blast_i_rad**2)
c
c Determine the total pressure on faces inside the
c blast wave
c
        blast_pressure = blast_energy / blast_volume
c
c Loop over all faces in the set
c
        do iface = 1, num_faces
c
c Extract the topology of the current face
c
        cur_face_id = faceID(iface)
        call aupst_get_face_topology(1, cur_face_id,
&                face_topo, query_error)
c
c Determine the number of nodes of the current face
c
        num_nodes = mod(face_topo,100)
c
c Extract the node ids for nodes contained in the current
c face
c
        call aupst_get_face_nodes(1, cur_face_id,
&                face_nodes, query_error)
c
c Extract the nodal coordinates of the face nodes
c
        call aupst_get_node_var(num_nodes, 3, face_nodes,
&                face_coords, "coordinates", query_error)
c
c Compute the centroid of the face
c
        face_center(1) = 0.0
        face_center(2) = 0.0

```

```

        face_center(3) = 0.0
        do inode = 1, num_nodes
            face_center(1) = face_center(1) +
&                face_coords(1,inode)
            face_center(2) = face_center(2) +
&                face_coords(2,inode)
            face_center(3) = face_center(3) +
&                face_coords(3,inode)
        enddo
        face_center(1) = face_center(1)/num_nodes
        face_center(2) = face_center(2)/num_nodes
        face_center(3) = face_center(3)/num_nodes
c
c Determine the distance from the current face
c to the blast center
c
        dist = sqrt((face_center(1) - pos_x)**2 +
&                (face_center(2) - pos_y)**2 +
&                (face_center(3) - pos_z)**2)
c
c Apply pressure to the current face if it falls within
c the blast wave
c
        if(dist .ge. blast_i_rad .and.
&        dist .le. blast_o_rad) then
            pressure(1,iface) = blast_pressure
        else
            pressure(1,iface) = 0.0
        endif
    enddo
    err_code = 0
end

```

11.4.2 Error Between a Computed and an Analytic Solution

The following code is a user subroutine to compute the error between Sierra/SM-computed results and results from an analytic manufactured solution. This subroutine is called by a `USER OUTPUT` command block immediately before producing an output Exodus file. The error for the mesh is computed by taking the squared difference between the computed and analytic displacements at every node. Finally, a global sum of the error is produced along with the square root norm of the error.

This user subroutine requires a user variable, which is defined in the Sierra/SM input file. The command block for the user variable specified in this user subroutine is as follows:

```

begin user variable conv_error
    type = global real length = 1

```

```

    global operator = sum
    initial value = 0.0
end user variable conv_error

```

The subroutine is called in the Sierra/SM input file as follows:

```

begin user output
  node set = nodelist_10
  node set subroutine = conv0_error
  subroutine real parameter: char_length = 1.0
  subroutine real parameter: char_time  = 1.0e-3
  subroutine real parameter: x_offset   = 0.0
  subroutine real parameter: y_offset   = 0.0
  subroutine real parameter: z_offset   = 0.0
  subroutine real parameter: t_offset   = 0.0
  subroutine real parameter: u0         = 0.01
  subroutine real parameter: v0         = 0.02
  subroutine real parameter: w0         = 0.03
  subroutine real parameter: alpha      = 1.0
  subroutine real parameter: youngs_modulus = 10.0e6
  subroutine real parameter: poissons_ratio = 0.3
  subroutine real parameter: density      = 0.0002588
  subroutine real parameter: num_nodes    = 125.0
end user output

```

The FORTRAN listing for the subroutine is as follows:

```

      subroutine conv0_error(num_nodes, num_vals,
&   eval_time, nodeID, values, flags, ierror)
      implicit none

      integer num_nodes
      integer num_vals
      double precision eval_time
      integer nodeID(num_nodes)
      double precision values(1)
      integer flags(1)
      integer ierror

c
c   Local vars
c
      integer inode
      integer error_code
      double precision clength, ctime, xoff, yoff, zoff, toff
      double precision zero, one, two, three, four, nine
      double precision mod_coords(3,3000)

```

```

double precision cdispl(3,3000)
integer num_comp_check
double precision expat
double precision x, y, z, t
double precision u0, v0, w0, alpha
double precision pi
double precision half
double precision mdisplx, mdisply, mdisplz
double precision xdiff, ydiff, zdiff
double precision conv_error
double precision numnod

pi      = 3.141592654
half    = 0.5
zero    = 0.0
one     = 1.0
two     = 2.0
three   = 3.0
four    = 4.0
nine    = 9.0

c
c Check that the nodal coordinates will fit into the
c statically allocated array
c
      if(num_nodes .gt. 3000) then
        write(6,*) ERROR in sphere disp, cdispl,
& num_nodes exceeds static array size
        ierror = 1
        return
      endif

c
c Extract the model coordinates for all nodes
c
      call aupst_check_node_var(num_nodes, num_comp_check,
&                               nodeID, "model_coordinates",
&                               ierror)
      if(ierror .ne. 0) return
      if(num_comp_check .ne. 3) return
      call aupst_get_node_var(num_nodes, num_comp_check,
&                             nodeID, mod_coords, "model_coordinates",
&                             ierror)

c
c Extract the computed displacements for all nodes
c
      call aupst_check_node_var(num_nodes, num_comp_check,
&                               nodeID, "displacement",
&                               ierror)

```

```

        if(ierr .ne. 0) return
        if(num_comp_check .ne. 3) return
        call aupst_get_node_var(num_nodes, num_comp_check,
&            nodeID, cdispl, "displacement",
&            ierr)
c
c Extract the subroutine parameters.
c
        call aupst_get_real_param("char_length",
&                                clength,error_code)
        call aupst_get_real_param("char_time",
&                                ctime,error_code)
        call aupst_get_real_param("x_offset",xoff,error_code)
        call aupst_get_real_param("y_offset",yoff,error_code)
        call aupst_get_real_param("z_offset",zoff,error_code)
        call aupst_get_real_param("t_offset",toff,error_code)
        call aupst_get_real_param("u0",u0,error_code)
        call aupst_get_real_param("v0",v0,error_code)
        call aupst_get_real_param("w0",w0,error_code)
        call aupst_get_real_param("alpha",alpha,error_code)
        call aupst_get_real_param("num_nodes",
&                                numnod,error_code)
c
c Calculate a solution scaling factor
c
        expat = half * ( one - cos( pi * eval_time / ctime ) )
c
c Compute the expected solution at each node and do a
c sum of the differences from the analytic solution
c
        conv_error = zero
        do inode = 1, num_nodes
c
c Set the displacement value from the manufactured solution
c
            x = ( mod_coords(1,inode) - xoff ) / clength
            y = ( mod_coords(2,inode) - yoff ) / clength
            z = ( mod_coords(3,inode) - zoff ) / clength
c
            mdisplx = u0 * sin(x) * cos(two*y) * cos(three*z)
            *
            * expat
            mdisply = v0 * cos(three*x) * sin(y) * cos(two*z)
            *
            * expat
            mdisplz = w0 * cos(two*x) * cos(three*y) * sin(z)
            *
            * expat
c
            xdiff = mdisplx - cdispl(1,inode)

```

```

        ydiff = mdisply - cdispl(2,inode)
        zdiff = mdisplz - cdispl(3,inode)
        conv_error = conv_error + xdiff*xdiff
        *                               + ydiff*ydiff
        *                               + zdiff*zdiff
c
        enddo
c
        ierror = 0
c
c Do a parallel sum of the squared errors and extract
c the total summed value on all processors
c
        call aupst_put_global_var(1,conv_error,
&                               "conv_error", "sum", ierror)
        call aupst_get_global_var(1,conv_error,
&                               "conv_error", ierror)
c
c Take the square root of the errors and store that as
c the net error norm
c
        conv_error = sqrt(conv_error) / sqrt(numnod)
        call aupst_put_global_var(1,conv_error,
&                               "conv_error", "none", ierror)
c
        return
        end

```

11.4.3 Transform Output Stresses to a Cylindrical Coordinate System

The following code is a user subroutine to transform element stresses in global x , y , and z coordinates to a global cylindrical coordinate system. This subroutine could be used to transform the relatively meaningless shell stress in x , y , and z coordinates to more meaningful tangential, hoop, and radial stresses. The subroutine is called from a `USER OUTPUT` command block. It reads in the old stresses, transforms them, and writes them back out to a user-created scratch variable, defined via a `USER VARIABLE` command block, for output.

```

begin user variable cyl_stress
    type = element sym_tensor length = 1
    initial value = 0.0 0.0 0.0 0.0 0.0 0.0
end user variable

begin user output
    block = block_1
    element block subroutine = aupst_cyl_transform
    subroutine string parameter: origin_point = Point_O
    subroutine string parameter: z_point      = Point_Z

```

```

subroutine string parameter: xz_point      = Point_XZ
subroutine string parameter: input_stress = memb_stress
subroutine string parameter: output_stress = cyl_stress
end user output

```

The FORTRAN listing for the subroutine is as follows:

```

subroutine aupst_cyl_transform(num_elems, num_vals,
*  eval_time, elemID, values, flags, ierror)
implicit none
#include<framework/Fmwk_type_sizes_decl.par>
#include<framework/Fmwk_type_sizes.par>
c
c  Subroutine Arguments
c
c  num_elems: Input: Number of elements to calculate on
c  num_vals  : Input: Ignored
c  eval_time: Input: Time at which to evaluate the stress.
c  elemID    : Input: Global sierra IDs of the input elements
c  values    : I/O   : Ignored, stress will be stored manually
c  flags     : I/O   : Ignored
c  ierror    :Output: Returns non-zero if an error occurs
c
c      integer num_elems
c      integer num_vals
c      double precision eval_time
c      integer elemID(num_elems)
c      double precision values(1)
c      integer flags(1)
c      integer ierror
c
c  Fortran cannot dynamically allocate memory, thus worksets
c  will be iterated over by chunks each of size chunk_size.
c
c      integer chunk_size
c      parameter (chunk_size = 100)
c      integer chunk_ids(chunk_size)
c
c  Subroutine parameter data
c
c      character*80      origin_point_name
c      double precision origin_point(3)
c      character*80      z_point_name
c      double precision z_point(3)
c      character*80      xz_point_name
c      double precision xz_point(3)
c      character*80      input_stress_name

```

```

        character*80      output_stress_name
c
c  Local element data for centroids and rotation vectors
c
        double precision cent(3)
        double precision centerline_pos(3)
        double precision dot_prod
        double precision z_vec(3)
        double precision r_vec(3)
        double precision theta_vec(3)
        double precision rotation_tensor(9)
c
c  Chunk data storage
c
        double precision elem_centroid(3, chunk_size)
        double precision input_stress_val(6, chunk_size)
        double precision output_stress_val(6, chunk_size)
c
c  Simple iteration variables
c
        integer error_code
        integer ichunk, ielem
        integer zero_elem, nel
c
c  Extract the current subroutine parameters.  origin_point
c  is the origin of the coordinate system
c  z_point is a point on the z axis of the coordinate system
c  xz_point is a point on the xz plane
c
        call aupst_get_string_param("origin_point",
&                                origin_point_name,
&                                error_code)
        call aupst_get_string_param("z_point",
&                                z_point_name,
&                                error_code)
        call aupst_get_string_param("xz_point",
&                                xz_point_name,
&                                error_code)
        call aupst_get_string_param("input_stress",
&                                input_stress_name,
&                                error_code)
        call aupst_get_string_param("output_stress",
&                                output_stress_name,
&                                error_code)
c
c  Use the point names to look up the coordinates of each
c  relevant point

```



```

c      call aupst_get_point(origin_point_name, origin_point,
c      &                      error_code)
c      call aupst_get_point(z_point_name, z_point,
c      &                      error_code)
c      call aupst_get_point(xz_point_name, xz_point,
c      &                      error_code)
c
c      Compute the z axis vector
c
c      z_vec(1) = z_point(1) - origin_point(1)
c      z_vec(2) = z_point(2) - origin_point(2)
c      z_vec(3) = z_point(3) - origin_point(3)
c
c      Transform z_vec into a unit vector, abort if it is invalid
c
c      call aupst_unitize_vector(z_vec, ierror)
c      if(ierror .ne. 0) return
c
c      Loop over chunks of the data arrays
c
c      do ichunk = 1, (num_elems/chunk_size + 1)
c
c      Determine the first and last element number for the
c      current chunk of elements
c
c      zero_elem = (ichunk-1) * chunk_size
c      if((zero_elem + chunk_size) .gt. num_elems) then
c        nel = num_elems - zero_elem
c      else
c        nel = chunk_size
c      endif
c
c      Copy the elemIDs for all elems in the current chunk to a
c      temporary array
c
c      do ielem = 1, nel
c        chunk_ids(ielem) = elemID(zero_elem + ielem)
c      enddo
c
c      Extract the element centroids and stresses
c
c      call aupst_get_elem_centroid(nel, chunk_ids,
c      &                              elem_centroid,
c      &                              ierror)
c      call aupst_get_elem_var(nel, 6, chunk_ids,
c      &                              input_stress_val,

```

```

&                                input_stress_name, ierror)
c
c Loop over each element in the current chunk
c
c     do ielem = 1, nel
c
c Find the closest point on the cylinder centerline axis
c to the element centroid
c
c     cent(1) = elem_centroid(1, ielem) - origin_point(1)
c     cent(2) = elem_centroid(2, ielem) - origin_point(2)
c     cent(3) = elem_centroid(3, ielem) - origin_point(3)
c     dot_prod = cent(1) * z_vec(1) +
&               cent(2) * z_vec(2) +
&               cent(3) * z_vec(3)
c     centerline_pos(1) = z_vec(1) * dot_prod
c     centerline_pos(2) = z_vec(2) * dot_prod
c     centerline_pos(3) = z_vec(3) * dot_prod
c
c Compute the current normal radial vector
c
c     r_vec(1) = cent(1) - centerline_pos(1)
c     r_vec(2) = cent(2) - centerline_pos(2)
c     r_vec(3) = cent(3) - centerline_pos(3)
c     call aupst_unitize_vector(r_vec, ierror)
c     if(ierror .ne. 0) return
c
c Compute the current hoop vector
c
c     theta_vec(1) = z_vec(2)*r_vec(3) - r_vec(2)*z_vec(3)
c     theta_vec(2) = z_vec(3)*r_vec(1) - r_vec(3)*z_vec(1)
c     theta_vec(3) = z_vec(1)*r_vec(2) - r_vec(1)*z_vec(2)
c
c The r, theta, and z vectors describe the new stress
c coordinate system, Transform the input stress tensor
c in x,y,z coords to the output stress tensor in r, theta,
c and z coords use the unit vectors to create a rotation
c tensor
c
c     rotation_tensor(k_f36xx) = r_vec(1)
c     rotation_tensor(k_f36yx) = r_vec(2)
c     rotation_tensor(k_f36zx) = r_vec(3)
c     rotation_tensor(k_f36xy) = theta_vec(1)
c     rotation_tensor(k_f36yy) = theta_vec(2)
c     rotation_tensor(k_f36zy) = theta_vec(3)
c     rotation_tensor(k_f36xz) = z_vec(1)
c     rotation_tensor(k_f36yz) = z_vec(2)

```

```

        rotation_tensor(k_f36zz) = z_vec(3)
c
c Rotate the current stress tensor to the new configuration
c
        call fmth_rotate_symten33(1, 1, 0, rotation_tensor,
&                                input_stress_val(1,ielem),
&                                output_stress_val(1,ielem))
        enddo
c
c Store the new stress
c
        call aupst_put_elem_var(nel, 6, chunk_ids,
&                                output_stress_val,
&                                output_stress_name, ierror)
        enddo
        ierror = 0
    end

```

11.5 User Subroutine Library

Many user subroutines are used commonly and have been permanently incorporated into the code. These subroutines are used just like any other subroutines, but they do not need to be manually compiled into the code. (The user need be concerned only about the Sierra/SM command lines.) Additionally the subroutines listed here are regularly tested with Sierra/SM and formally maintained by the development team.

This section describes the usage of each of these user subroutine library subroutines..

11.5.1 aupst_preload_solver

Author: Nathan Crane

Note, use of the 'aupst_preload_solver' subroutine is also documented in the example manual chapter 'Automated Adaptive Preloading'.

Purpose:

This subroutine is used to solve simple inverse problems for preload conditions determining the value of a working variable to achieve some target condition. Examples where this subroutine has been successfully used include: apply the target strain in bolts to yield specified clamping force in each bolt, find the force that must be applied to a pin to reach a target displacement in that pin, and to find the displacement at a fixture to yield a target contact force on a body.

There are a number of restrictions on the relations that can be effectively solved by this subroutine. The first is that the relationship is monotonic. For example adding more strain to bolt increases the clamping force. If the relationship is not monotonic the solution may stall at a local maximum. The second restriction is that the relationship is at least somewhat smooth. The relationship between changes in the working variable to the target condition does not need to be linear, but should be at least nominally C0 continuous. Third the relationship must be somewhat strong. Again as an example changing the artificial strain in a bolt strongly affects its clamping force.

The subroutine itself is generally called by a `USER OUTPUT` command block and often needs to be combined with a number of other commands as shown in this example:

```
# Drive element preload as a variable set by
# aupst_preload_solver subroutine
begin function bolt_preload
  type is analytic
  expression variable: v = element applied_strain
  evaluate expression = "v"
end
# Internal variables needed by subroutine
begin user variable applied_strain
  type = element real length = 1
  use with restart
end
begin user variable bolt_preload_state
```

```

    type = element real length = 12
    use with restart
end
# Use the 'working' variable to apply strain to bolt
begin artificial strain
    block = block_201
    r function = sierra_constant_function_zero
    s function = sierra_constant_function_zero
    t function = bolt_preload
end
# Compute the currently obtained clamping force
# on the bolt head.
begin user output
    surface = surface_200
    compute global bolt_force as sum of nodal force_contact(z)
end
# Solver for preload. Update the working variable to try
# to make the target_variable equal to the target value
begin user output
    block = block_201
    subroutine real parameter: target_value      = 1.0e+7
    subroutine real parameter: initial_guess     = -3.06e-4
    subroutine real parameter: iteration_time    = 5.0e-5
    subroutine string parameter: target_variable = global bolt_force
    subroutine string parameter: working_variable = element applied_strain
    subroutine string parameter: state_variable  = element bolt_preload_state
    element block subroutine = aupst_preload_solver
    compute at every step
end

```

Parameters:

The meaning of some of the parameters are shown visually in Figure 11.2. For the bolt preload example the target artificial strain starts low and monotonically asymptotes up to the correct solution over several predictor corrector iteration cycles.

Subroutine Operation:

The subroutine drives a predictor correct iteration in time. Each predictor corrector cycle is performed over the input deck defined 'iteration_time'. The step cycle is shown in Figure 11.3 the value of the working variable produced by the subroutine is shown in blue and an example value of the response of the target variable shown in yellow. During the first and second quarter of the cycle the working variable is slowly increased from its old value to the new value. This ramp up is done with a smooth half cycloidal loading ramp. On the first cycle this new value will be half of the initial guess. During the third quarter of the cycle the working variable is held constant. Ideally damping terms will settle the value of the target variable to some relatively constant value during this time. In the fourth quarter of the load cycle the working variable is again held constant and the target variable is evaluated. The evaluation is the time average of the target variable over this

Parameter Name	Type	Description
target_value	Real	The value that the target_variable is trying to reach. In this example the target clamping force in the bolt
initial_guess	Real	An initial guess for the working_variable. In this example the initial guess for artificial strain.
iteration_time	Real	Amount of analysis time each predictor-corrector iteration takes
target_variable	String	Type and name of the target_variable. In this example the current clamping force in the bolt as computed by the sum of contact force on the bolt head.
working_variable	String	Type and name of the working_variable. In this example the artificial strain magnitude applied to each element of the bolt
state_variable	String	Type and name of state variables used internally by the subroutine. State variable type must match working variable type.

Table 11.27: Required aupst_preload_solver Parameters

fourth of the cycle time. Finally, at the end of the cycle, the new value for the next cycle working variable is defined based on the tangent of the response from the previous to the current iteration. The new working variable value is set such that the error in the target variable versus the target value will be roughly halved. This one half safety factor is used in an attempt to converge the solution monotonically from below. Additionally, regardless of the function tangent computed, the working variable can change by no more than the initial guess each iteration. This is an additional safety factor to prevent overshooting the solution.

It is important to consider that these predictor-corrector iterations only make sense if the target variable reaches a reasonably stable value by the fourth part of each iteration in order to make a reliable next prediction. For explicit transient dynamics these preload iterations should usually encompass several thousand time steps, but the actual number of steps needed may vary.

The preload solver subroutine will also operate with implicit statics (adagio). One consideration in this case is that each preload iteration cycle will take a minimum of three time steps: a ramp-up step, a hold step, and an evaluation step.

State Variables:

The subroutine state variables are used primarily for the internal operation of the subroutine. These variables can be also be post processed to understand how the subroutine is operating. The description of each variable is shown in the below Table 11.28. Depending on how the subroutine is used these state variables could be stored in a global variable or there could be a set of state variables at each node or element that is being loaded. The twelfth state variable, which indicates the current

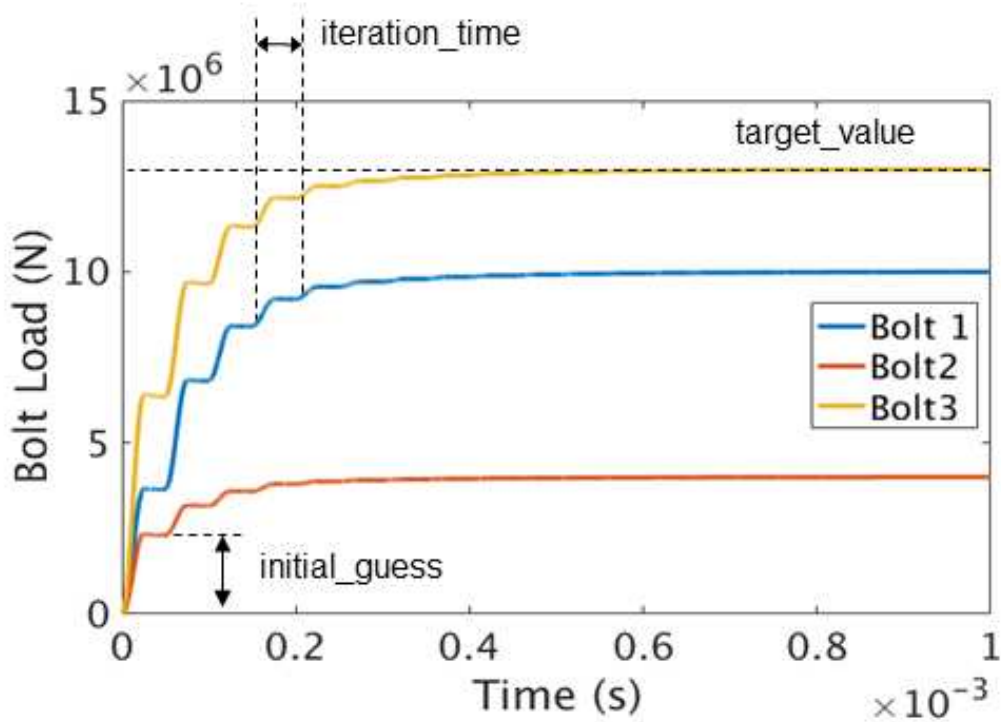


Figure 11.2: Example of Preload Iteration

error in the preload solution can be particularly useful to look at.

Usage Guidelines and Warnings:

Pay special care to the 'use with restart' and 'compute at every step' commands present in the example input blocks. The 'use with restart' flag ensures the state variables are properly stored in any restart database ensuring the subroutine works correctly with a restart analysis. The 'compute at every step' command ensures the preload solver correctly updates each an every step. Due to its presence in the 'user output' command block without this command the preload solver would only evaluate at time steps where output is performed, likely yielding very strange results in the analysis.

The iteration time must be long enough that the target variable can re-equilibrate to some relatively constant value in time in the target variable evaluation part of the cycle. The subroutine will compare previously found target variable values to current target variable values to make a new prediction for the working variable. If the target variable has a large dynamic oscillation that causes its value to change unrelated to the working variable change this prediction will be poor and the solver will not function correctly.

The analysis must run for enough time that a sufficient number of predictor corrector preload cycles are taken to converge the target variable to the target value. How many cycles are required will

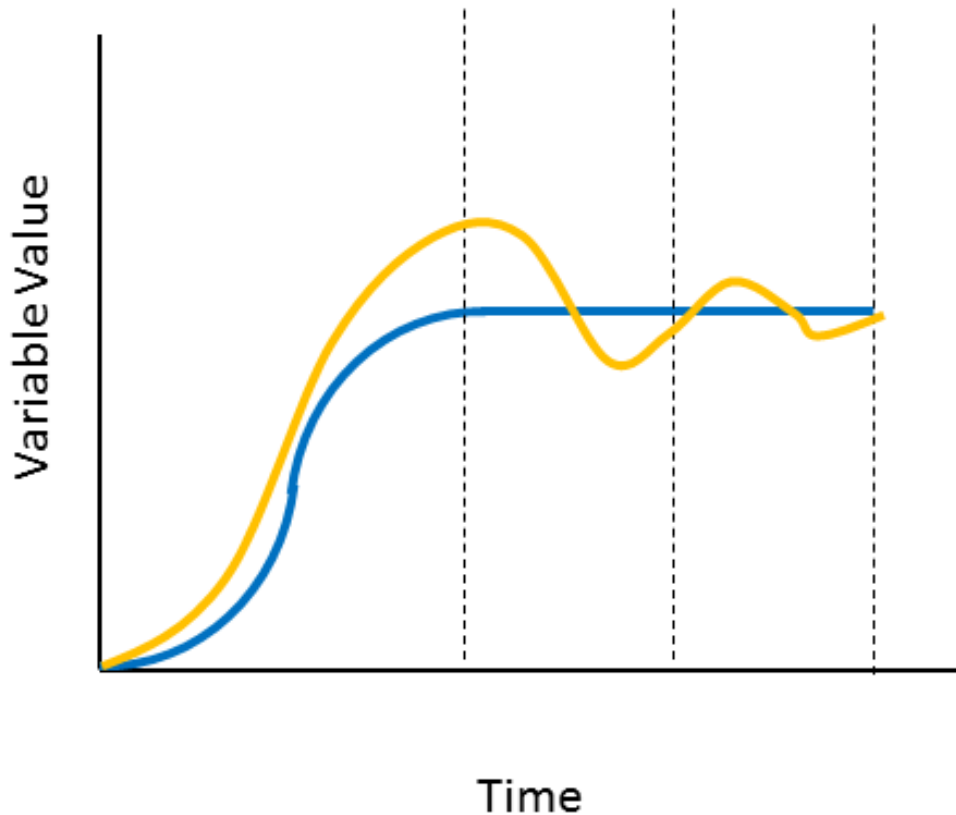


Figure 11.3: Anatomy of the Preload Iteration Cycle

depend on how close the initial guess is. However, due to conservative assumptions in the solver usually at least 10 cycles will be required even if the initial guess is good.

This subroutine uses a simple predictor corrector approach and can only solve for conditions where the target vs. working variable response is both monotonic and relatively smooth (i.e., at least C^0 continuous).

It is strongly recommended that results be investigated both to ensure an acceptable error bound has been reached and that a reasonable history of the working variable was applied. For example greatly overshooting the target on the initial guess could cause spurious yield which would then be locked into the solution.

A few different combinations of variable types may be used. The working and state variable must be the same type. The target variable must be either a global or the same type as the working and state variables.

State Number	Description
1	Start time of current preload iteration
2	End time of current preload iteration
3	Value of working variable at start of current preload iteration
4	Value of working variable at end of current preload iteration
5	Iteration Phase. 1=ramp up (1st and 2nd quarter), 2=hold (3rd quarter), 3=hold/evaluate (4th quarter)
6	Variables 6 and 7 are used to compute a running average of the target variable during the hold/evaluate phase
7	
8	Initial (very first time) value of the working variable
9	Initial (very first time) value of the target variable
10	The time the subroutine was last evaluated
11	The target variable that was computed in the previous preload iteration
12	The relative solution error in target value vs. variable error starts at 1.0 and ideally reaches 0.0

Table 11.28: aupst_preload_solver Subroutine State Variables

11.5.2 aupst_torque_calc

Author: Nathan Crane

Purpose:

This subroutine computes the net torque applied to a body by a set of forces. The torque is defined around an axis defined by two points. The output torque is given as a single value about this axis with a sign following the right hand rule. This subroutine is generally called by a USER OUTPUT command block. For example:

```

begin user output
  node set = nodelist_1
  compute global ns1 as average of nodal coordinates
end
begin user output
  node set = nodelist_2
  compute global ns2 as average of nodal coordinates
end
begin user variable moment1
  type = global real length = 1
  global operator = sum
end
begin user output

```

```

surface = surface_1      #Contains nodes over which to do
                        # moment summation
node set subroutine = aupst_torque_calc
subroutine string parameter: axis_node_1 = ns1
subroutine string parameter: axis_node_2 = ns2
subroutine string parameter: force_field = reaction
subroutine string parameter: output_var = moment1
end

```

Parameters:

Parameter Name	Type	Description
axis_node_1	String	Name of global variable containing location of first axis node
axis_node_2	String	Name of global variable containing location of second axis node
force_field	String	Name of field containing force values to sum into the torque
output_var	String	Name of output global variable to store the computed moment

Table 11.29: Required aupst_torque_calc Parameters

11.5.3 aupst_trans_rot_velocity

Author: Nathan Crane

Purpose:

This subroutine sets an initial velocity on a part that is the combination of an arbitrary rigid body spin rate superimposed on an arbitrary translational velocity. This subroutine is generally called by a INITIAL VELOCITY command block. For example:

```

begin initial velocity
  block = block_2
  node set subroutine = aupst_trans_rot_velocity
  subroutine real parameter: center_x = 0.0
  subroutine real parameter: center_y = 0.0
  subroutine real parameter: center_z = 2.0

  subroutine real parameter: rot_dir_x = 1.0
  subroutine real parameter: rot_dir_y = 1.0
  subroutine real parameter: rot_dir_z = 1.0

  subroutine real parameter: angular_velocity = 628.31853

```

```

subroutine real parameter: tran_dir_x = 1.0
subroutine real parameter: tran_dir_y = 1.0
subroutine real parameter: tran_dir_z = 1.0

subroutine real parameter: translational_velocity = 173.20508
end

```

Parameters:

Parameter Name	Type	Description
center_x, center_y, center_z	Real	Coordinates of center of rotation
rot_dir_x, rot_dir_y, rot_dir_z	Real	Direction of axis of rotation
tran_dir_x, tan_dir_y, tan_dir_z	Real	Direction of translational velocity
angular_velocity	Real	Spin rate in radians per time
translational_velocity	Real	translational velocity in distance per time

Table 11.30: Required aupst_trans_rot_velocity Parameters

11.5.4 aupst_cyl_transform

Author: Nathan Crane

Purpose:

This subroutine transforms element stresses from a global rectangular coordinate system to a local cylindrical coordinate system. This subroutine is generally called by a `USER OUTPUT` command block. For example:

```

begin user output
  block = block_1
  element block subroutine = aupst_cyl_transform
  subroutine string parameter: origin_point = Point_O
  subroutine string parameter: z_point      = Point_Z
  subroutine string parameter: xz_point     = Point_XZ
  subroutine string parameter: input_stress = memb_stress
  subroutine string parameter: output_stress = cyl_stress
end user output

```

Requirements:

This subroutine requires a tensor variable to store the cylindrical stress into a variable for each element. The variable is created by the following command block in the Sierra/SM region:

```
begin user variable cyl_stress
  type = element sym_tensor length = 1
  initial value = 0.0 0.0 0.0 0.0 0.0 0.0
end user variable
```

Parameters:

Parameter Name	Usage	Description
origin_point	String	Name of the point at the cylinder origin.
z_point	String	Point on the cylinder axis.
xz_point	String	Point on the line that passes through theta = 0 on the cylinder.
input_stress	String	Name of the Sierra/SM internal input stress tensor variable.
output_stress	String	Name of the Sierra/SM internal output stress tensor variable.

11.5.5 aupst_rec_transform**Author:** Daniel Hammerand**Purpose:**

This subroutine transforms element stresses from a global rectangular coordinate system to a different local rectangular coordinate system. This subroutine is generally called by a `USER OUTPUT` command block. For example:

```

begin user output
  block = block_1
  element block subroutine = aupst_rec_transform
  subroutine string parameter: origin_point = Point_O
  subroutine string parameter: z_point      = Point_Z
  subroutine string parameter: xz_point     = Point_XZ
  subroutine string parameter: input_stress = memb_stress
  subroutine string parameter: output_stress = new_stress
end user output

```

Requirements:

This subroutine requires a tensor variable to store the new stress into a variable for each element. The variable is created by the following command block in the Sierra/SM region:

```

begin user variable new_stress
  type = element sym_tensor length = 1
  initial value = 0.0 0.0 0.0 0.0 0.0 0.0
end user variable

```

Parameters:

Parameter Name	Usage	Description
origin_point	String	Name of the point at the cylinder origin.
z_point	String	Point on the cylinder axis.
xz_point	String	Point on the line that passes through theta = 0 on the cylinder.
input_stress	String	Name of the Sierra/SM internal input stress tensor variable.
output_stress	String	Name of the Sierra/SM internal output stress tensor variable.

11.5.6 copy_data**Author:** Jason Hales**Purpose:**

This subroutine copies data from one variable to another with offsets given for both variables. This subroutine is generally called by a `USER OUTPUT` command block. For example:

```

begin user output
  block = block_1
  element block subroutine = copy_data
  subroutine integer parameter: source_offset = 4
  subroutine string parameter:  source_name = stress
  subroutine integer parameter: destination_offset = 1
  subroutine string parameter:  destination_name = uservarxy
end user output

```

Requirements:

This subroutine requires that the source and destination fields exist and have lengths at least as great as the values supplied as offsets. The fields used may be defined by the user as variables. In this example, the variable is created by the following command block in the Sierra/SM region:

```

begin user variable uservarxy
  type = element real length = 1
  initial value = 0.0
end user variable

```

Parameters:

Parameter Name	Usage	Description
source_offset	Integer	The offset into the source variable.
source_name	String	The name of the source variable.
destination_offset	Integer	The offset into the destination variable.
destination_name	String	The name of the destination variable.

11.5.7 trace

Author: Jason Hales

Purpose:

This subroutine computes the trace of a tensor. This subroutine is generally called by a USER OUTPUT command block. For example:

```
begin user output
  block = block_1
  element block subroutine = trace
  subroutine string parameter: source_name = log_strain
  subroutine string parameter: destination_name = uvarbulkstrain
end user output
```

Requirements:

This subroutine requires that the source and destination fields exist. The source field should have a length of six. The destination field should have a length of one. The destination field will typically be defined by the user as a variable. In this example, the variable is created by the following command block in the Sierra/SM region:

```
begin user variable uvarbulkstrain
  type = element real length = 1
  initial value = 0.0
end user variable
```

Parameters:

Parameter Name	Usage	Description
source_name	String	The name of the source variable.
destination_name	String	The name of the destination variable.

11.5.8 aupst_cavity_volume**Author:** Nathan Crane**Purpose:**

Compute the volume of an enclosed side set. This subroutine is generally called by a USER OUTPUT command block. For example:

```
begin user output
  SURFACE = surface_4000
  SURFACE SUBROUTINE = aupst_cavity_volume
  subroutine string parameter: outputVarName = s4000_vol

  SUBROUTINE REAL PARAMETER: cavitySymmetryPoint_x = 0.0
  SUBROUTINE REAL PARAMETER: cavitySymmetryPoint_y = 0.0
  SUBROUTINE REAL PARAMETER: cavitySymmetryPoint_z = 0.0
  SUBROUTINE REAL PARAMETER: volumeScale           = 2.0

  compute at every step
end
```

Availability:

Compiled into Adagio 4.24 and later.

Compatible with Presto 2.2 / Adagio 2.2 and later.

Parameters:

Name	Type	Description
outputVarName	String	Name of global variable in which to store the computed volume.
cavitySymmetryPoint_x	Real	Optional: X coordinate of a point on the intersection of all model symmetry planes. Needed if the cavity is not fully enclosed due to open symmetry planes.
cavitySymmetryPoint_y	Real	Optional: Y coordinate of a point on the intersection of all model symmetry planes. Needed if the cavity is not fully enclosed due to open symmetry planes.
cavitySymmetryPoint_z	Real	Optional: Z coordinate of a point on the intersection of all model symmetry planes. Needed if the cavity is not fully enclosed due to open symmetry planes.
volumeScale	Real	Optional: Value to multiple the final cavity volume by. Defaults to 1.0. May be used to adjust for symmetry boundary conditions.

Chapter 12

Transfers

It is sometimes desirable to chain two or more analyses (procedures) together. A common example of this is the need to preload an assembly quasistatically, then subject that assembly to a loading environment best suited to an explicit transient dynamics analysis. The displacements and stresses produced by the quasistatic preload become initial conditions for the transient dynamics event. These displacements and stresses must therefore be transferred from the initial to the subsequent analysis.

This chapter reviews the concept of transfers in SIERRA and outlines the syntax required to perform a transfer of information between procedures.

12.1 SIERRA Transfers

Applications built on the SIERRA computational framework share underlying data structures. This makes it convenient to couple applications together using transfers.

The coupling available through SIERRA is of two types. The first is what is called intra-procedural coupling. In this case, multiple regions within a single procedure share data. This enables multi-physics analysis such as thermal-mechanical coupling. The details of this type of coupling, along with the syntax to support it, will not be covered here. Coupled codes such as Arpeggio use this type of coupling.

The second type of coupling is inter-procedural coupling. Here, the result from one procedure is handed to the next procedure. This is the type of coupling used when moving from one analysis stage to another in Adagio and Presto. When using this type of coupling, the two procedures generally have only one region each.

12.2 Inter-procedural Transfers

The inter-procedural transfers used by Adagio and Presto can transfer data from one or all blocks of the preceding or sending procedure to the subsequent or receiving procedure. The commands to control the transfers should appear at the procedure scope in the receiving procedure.

When using inter-procedural transfers, irrespective of the transfer type chosen, all global variables are copied by default from the sending procedure to the receiving procedure, and all appropriate element data is transferred from the sending to the receiving elements. Nodal data is transferred based on the type of analysis done in the two procedures. For example, if the first procedure is a quasistatic analysis and the second is an explicit transient dynamics analysis, the displacements of the nodes will be transferred but not their velocities.



Warning: Transfer of data for node-based tetrahedra is currently not fully supported. A coupled analysis with node-based tetrahedra will only be correct if the tetrahedral elements are not deformed in the first (sending) procedure, in which case initialization in the second (receiving) procedure is appropriate.

The available commands are given below.

```
BEGIN PROCEDURAL TRANSFER <string>name

BLOCK = <string list>block_name
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_name

ADDITIONAL ELEMENT VARIABLES = <string list>addElemVar
ELEMENT VARIABLES = <string list>elemVar
ELEMENT LIE VARIABLES = <string list>elemLieVar
NODE VARIABLES = <string list>nodeVar
GLOBAL VARIABLES = <string list>globalVar

ABSOLUTE SEARCH TOLERANCE = <real>tolerance (1.0e-5)

BEGIN INTERPOLATION|L2_PROJECTION TRANSFER <string>name

BLOCK BY BLOCK
NEAREST ELEMENT COPY

SEND BLOCKS = <string list>block_name
SEND COORDINATES = ORIGINAL|CURRENT (ORIGINAL)

RECEIVE BLOCKS = <string list>block_name
RECEIVE COORDINATES = ORIGINAL

TRANSFORMATION TYPE = NONE|RIGIDBODY|
REFLECT|ELEMENT2NODES|ELEMENT2ELEMENT
```

```

    LINEAR SOLVER = <string>solver_name(DEFAULT_FETI_SOLVER)

    END [ INTERPOLATION|L2_PROJECTION TRANSFER <string>name ]
    END [PROCEDURAL TRANSFER <string>name]

```

If the *same* finite element model is used in the sending and receiving regions, data may be *copied* directly from the sending to the receiving region. In this case, use of the BLOCK, INCLUDE ALL BLOCKS, and REMOVE BLOCK command lines alone will suffice to copy data for all blocks except those that are rigid bodies.

If *different* finite element models are used in the sending and receiving regions, or it is desired to transfer data for rigid bodies, an INTERPOLATION TRANSFER or L2_PROJECTION TRANSFER block must be used, and the the BLOCK, INCLUDE ALL BLOCKS, and REMOVE BLOCK command lines in the PROCEDURAL TRANSFER block should *not* be used. If these commands, appropriate only for the direct *copy* of data, are used when not applicable, their behavior is undefined.

12.2.1 Copying Data with Inter-procedural Transfers

To copy data from one or more blocks in a sending region to the matching blocks in the receiving region, use these line commands:

```

BLOCK = <string list>block_name
INCLUDE ALL BLOCKS
REMOVE BLOCK = <string list>block_name

```

The first of these, the BLOCK line command, is used to list blocks that should be copied. If it is desired to copy data for all blocks, use the INCLUDE ALL BLOCKS line command. Finally, one or more blocks may be removed from the set of all blocks with the REMOVE BLOCK line command. Either the BLOCK or INCLUDE ALL BLOCKS line command must appear for copy transfers.

12.2.2 Selecting Variables for Transfer

By default, the code determines the variables to transfer: all global variables, and the sets of node and element variables necessary to initialize a new procedure where the previous procedure finished. Additional variables may be added to the default set with the ADDITIONAL ELEMENT VARIABLES command.

Alternatively, a manual subgroup of variables may be transferred with a combination of the ELEMENT VARIABLES, ELEMENT LIE VARIABLES, NODE VARIABLES, and GLOBAL VARIABLES. If any of these commands are used then all default variable transfers will be deactivated and all desired transfer variables *must be manually specified*.

The ELEMENT LIE VARIABLES designation indicates that variables should undergo a specialized mapping for interpolation to preserve their spaces. Specifically, a logarithmic mapping of the scalar, vector, or tensor field is performed before any transfer operation is initiated. The logarithmically mapped fields are transferred and interpolated to the elements of the receiving mesh, after

which the values are exponentially mapped. This transfer method is more accurate, and appropriate for only variables that belong to Lie groups. In practice, these variables include rotations, deformation gradients, and stretches. The `ELEMENT LIE VARIABLES` option is valid only when an L_2 projection transfer is specified.

12.2.3 Additional Transfer Options

Additional procedural transfer options include the ability to specify the `absolute search tolerance`. The `absolute search tolerance` is the absolute amount that the search bounding box should be expanded in each of the six global directions (+/- x, y, and z) for variable transfers. This tolerance is applied to the sending region to determine which sending mesh objects (nodes or elements) contain, or are nearby, each receiving mesh object. If the bounding box search returns zero or multiple sending mesh objects, then a more expensive inverse isoparametric mapping search routine is invoked. The default `absolute search tolerance` is $1.0e-5$.

12.2.4 Interpolating Data with Interpolation Transfers

When different finite element models are used by the sending and receiving regions, or when it is desired to transfer data for rigid bodies, the `INTERPOLATION TRANSFER` or `L2_PROJECTION TRANSFER` must be used. The `INTERPOLATION` transfer uses a local interpolation patch. The `L2_PROJECTION` transfer relies on extrapolation of element fields to nodes and interpolation of those fields back to elements, which involves solving a global minimization problem. Generally, the `L2_PROJECTION` transfer is more accurate. For both the `INTERPOLATION` and `L2_PROJECTION` transfers, nodal variables are interpolated from the sending mesh to the receiving mesh.

The `TRANSFORMATION TYPE` command has the following options:

NONE (default) Do interpolation with no special transformation.

RIGIDBODY Transfer the rigid body information of each rigid body separately. Specify one and only one block in the `SEND BLOCKS` and `RECEIVE BLOCKS` line commands.

WEDGE23D Transform from a sending axisymmetric wedge model to a receiving full 3D cylinder model.

REFLECT Transform from a half-symmetric sending model to a full receiving model.

ELEMENT2NODES Usable only in the `L2_PROJECTION` transfer block. Project element variables on the sending model to nodal variables on the receiving model.

ELEMENT2ELEMENT Usable only in an `L2_PROJECTION` transfer block. Project element variables from the elements to the nodes of the sending model, transfer these projected nodal values from the sending to the receiving model via the normal interpolation transfer, then interpolate these nodal values to element variables on the receiving model. For continuous fields, this is generally the most accurate transfer method. This transfer type is combined with Lie algebra for variables belonging to Lie groups to preserve spaces of such variables.

When transferring data for non-rigid body blocks, the only required line commands are the `SEND BLOCKS` and `RECEIVE BLOCKS` line commands. These each list one or more blocks to be included in the transfer.

Interpolation transfers move data from sending to receiving meshes by performing searches and interpolating values using shape functions. Interpolation transfers will give the most accurate results when the sending and receiving meshes are similar. If the sending and receiving meshes do not occupy the same 3D space, for instance, transferred data is likely to be highly inaccurate.

In some instances, the sending and receiving meshes are highly similar such that there is a one-to-one correspondence between the list of sending blocks and the list of receiving blocks. In other words, it may be desired to send data from a given block to a corresponding block, from another block to its pair, and so forth. If this is the case, use of the `BLOCK BY BLOCK` line command will cause a separate transfer to be created for each pair of sending and receiving blocks. This is useful to ensure that data from a single block will be sent to one and only one receiving block. This could also be accomplished by listing multiple transfer blocks in the input file.

For certain use cases, such as remeshing for large deformations, it is desired to remesh on the deformed (current) configuration and adopt that as the new reference (model) configuration. For these use cases, use the command `SEND COORDINATES = CURRENT` instead of the default `SEND COORDINATES = ORIGINAL` so that the current configuration of the sending mesh is mapped to the model configuration of the receiving mesh when performing the search used by the transfers. Displacements are not transferred for this use case since deformations are assumed to be accounted for in the receiving mesh, i.e., the mesh was generated on the deformed configuration).



Warning: Adoption of a new reference configuration on the receiving mesh is currently only compatible with midpoint strain increment element formulations and hypo-elastic material models. All other element formulations and hyper-elastic materials require a multiplicative decomposition of the deformation gradient: the deformation gradient mapped from the sending mesh pre-multiplied by the deformation gradient due to the subsequent deformation applied on the receiving mesh. Such a multiplicative decomposition is not currently supported in transfers at this time.

12.2.4.1 Nodal variable interpolation transfer

The location of each receiving node in the receiving mesh is used to find a corresponding element in the sending mesh. The parametric coordinates of the coordinates associated with the receiving node within (or near) the sending element are calculated and subsequently used in conjunction with the element shape functions to interpolate the sending element nodal data onto each receiving node.

12.2.4.2 Element variable interpolation transfer

In an `INTERPOLATION TRANSFER` block, the center of each receiving element will be located within (or near) the closest element in the sending mesh. A patch of elements in the sending mesh

centered around the element containing the receiving element center is created. The receiving element value is then computed by interpolating the data in this patch of sending elements, using a least squares approach, to the point associated with the receiving element.

When the `NEAREST ELEMENT COPY` line command is used, data is sent directly from the closest sending element to the receiving element, forgoing the patch construction and least squares interpolation. A `NEAREST ELEMENT COPY` transfer is expected to be faster but potentially less accurate than an `INTERPOLATION TRANSFER`.

12.2.4.3 Element variable L_2 projection transfer

In an `L2_PROJECTION` transfer block, element variables at the numerical quadrature points of the sending mesh are first projected, via a global least squares solution, to the nodes of the sending mesh. A nodal interpolation transfer (described above) is then performed to transfer data from the nodes of the sending mesh to nodes of the receiving mesh. Finally, nodal data on the receiving mesh is interpolated to the quadrature points of the receiving mesh. A L_2 projection transfer is expected to yield more accurate element fields than an interpolation transfer.

For each block specified in `SEND BLOCKS`, a separate linear system, based on the consistent mass matrix (refer to Section 9.3.2.5 for details), is assembled and solved. Each element variable is projected to a nodal variable unique to the sending block so that potentially discontinuous fields at neighboring block boundaries are handled consistently.

In L_2 projection transfers based on higher-order element topologies such as ten-noded tetrahedra, the topology is transformed into an equivalent linear topology for extrapolation. Extrapolated nodal values are then linearly interpolated to the high-order nodes. This strategy has been shown to avoid spurious oscillations in the extrapolated quantities.

A particular linear solver may be selected for the L_2 projection using the `LINEAR SOLVER` user output command. The default is “`DEFAULT_FETI_SOLVER`.” Care should be taken to specify appropriate solver parameters and tolerances for the desired accuracy of the projection. Loose tolerances can significantly degrade the mapped solution. Generally, transfers consume a small percentage of the total run time, so a tight residual tolerance (e.g., 10^{-12}) is recommended.

12.3 Automated Implicit/Explicit Handoff

Explicit/Implicit hand off analyses are a common use case for Sierra/SM. For example, a component may be preloaded in an implicit quasistatic procedure; then, the preloaded component is subjected to an impact loading in an explicit dynamics procedure. Multi procedure analysis may be setup manually by defining multiple analysis regions, procedures, time controls, and transfer blocks. However, this manual process is time consuming and error prone.

The same functionality described above can be accomplished with the much simpler syntax below. The following example time block will perform an implicit to explicit hand off analysis.

```
BEGIN SM PROCEDURE myProcedure
  BEGIN TIME CONTROL
    BEGIN TIME STEPPING BLOCK preload
      START TIME = 0.0
```



```

    BEGIN PARAMETERS FOR SM REGION myRegion
      NUMBER OF TIME STEPS = 5
      TIME INTEGRATION = implicit_statics
      ADVANCE ON SOLVER FAILURE = current_procedure
    END
  END
BEGIN TIME STEPPING BLOCK impact
  START TIME = 0.1
  BEGIN PARAMETERS FOR SM REGION myRegion
    TIME INTEGRATION = explicit_dynamics
  END
END
termination time = 0.1001
END
BEGIN SM REGION myRegion
  ...Include region commands as normal here...
END
END

```

The automatic time period capability is triggered by using the `BEGIN SM PROCEDURE` and `BEGIN SM REGION` command lines rather than using `BEGIN PRESTO REGION` or `BEGIN ADAGIO REGION` procedures and regions. When these SM commands are used, the input deck is automatically run through a pre-processor to turn the single region multi time period input deck into an equivalent multi-procedure analysis. This preprocessed input deck is then automatically run by the sierra script. If the input deck originally has a name such as 'plateImpact.i' the preprocessed input deck will be saved to 'plateImpact.sm.i'. The modified input deck will also be echoed in the run log file.

The input deck preprocessing currently does the following operations to aid in setting up multiple region analysis.

- Time periods with `TIME_INTEGRATION` of `implicit_statics` will be converted into implicit adagio regions. Time periods with the `TIME_INTEGRATION` of `explicit_dynamics` will be converted into explicit presto regions. The appropriate begin and end times for each procedure and region will be automatically determined from the time period blocks defined in the 'sm' region. Additionally, the appropriate transfer operations between the regions will be automatically defined. The analysis procedures may switch back and forth between time integration schemes as many times as needed.
- The optional `ADVANCE ON SOLVER FAILURE` command can be used within an implicit analysis to define the course of action if the solver fails. The `current_procedure` option will kill the current procedure and start the next procedure using the last converged state of the solution. If the `current_time_period` option is used, the simulation will continue to the next time stepping block of the same procedure (if present). Keep in mind that the next time stepping block should not have a `START TIME`, since it will likely conflict with the last converged time of the previous time stepping block. The default option is `no_advancement`, which will terminate the simulation upon solver failure.
- Boundary conditions will be appropriately placed in each region based on their active peri-

ods.

- Each solution procedure will define its own results, history, and restart files. Output database names for results, history, and restart files will be automatically prepended with time period names to be unique for each analysis region. This file name modification prevents the output from one region overwriting the output of a previous region. If in the above example a results output file in the `sm` region is given the name `'myResult.e'` the analysis will output two separate results files `'preload_myResult.e'` and `'impact_myResult.e'`.
- If solution termination is used to dynamically swap between time periods and procedures these swaps are setup to function correctly with the multi-procedure analysis during the pre-processing.

The `SM` region preprocessing assumes all solution procedures use the same mesh, blocks, and material definitions. Additionally, transfers are always setup to transfer all possible analysis state data from one solution procedure to the next. More complex multi-region analyses cases can always be setup manually. For such manual setups, the automatically generated preprocessed `'sm'` input deck may still provide a good initial starting point for setting up a more complex input deck.

12.3.1 Known Issues and Limitations

Only capabilities that can be used in both implicit and explicit runs may be present in the `SM` region. For example this precludes use of `SPH` particles as this element formulation is only available in explicit analyses.

Handoffs using an `SM` region cannot be run coupled to another physics code through solution control. An example would be an arpeggio (aria/adagio) coupled analysis.

Appendix A

Explicit Dynamic Example Problem

This chapter provides an example problem to illustrate the construction of an input file for an analysis. The example problem consists of 124 spheres made of lead enclosed in a steel box. The steel box has an open top into which a steel plate is placed (see Figure A.1). A prescribed velocity is then applied on the steel plate, pushing it into the box and crushing the spheres contained within using frictionless contact. This problem is a severe test for the contact algorithms as the spheres crush into a nearly solid block. See Figure A.2 for results of this problem.

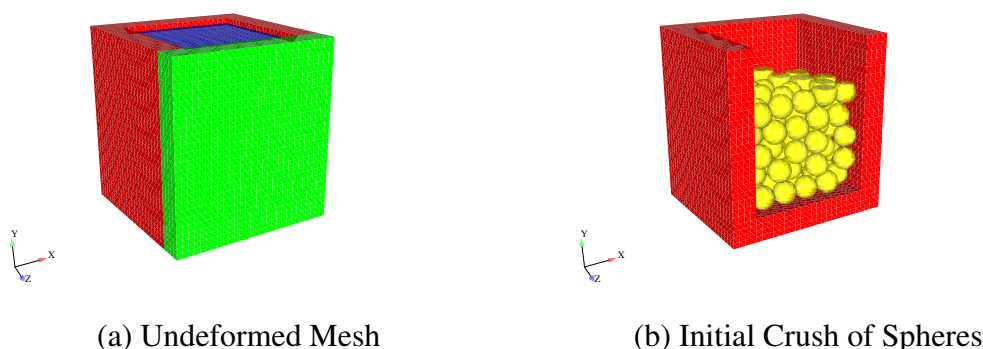


Figure A.1: Mesh for example problem: (a) box (red and green surfaces) with plate in top (blue surface) and (b) mesh with blue and green surfaces removed to show internal spheres (yellow) with initial crush.

The input file is described below, with comments to explain every few lines. Following the description, the full input file is listed again. Most of the key words in this example are all lowercase, which is different from the convention we have used to describe the command lines in this document. However, all the lowercase usage in the next example is an acceptable format in Presto.

The input file starts with a `begin sierra` statement (i.e., the first line of the `SIERRA` command block), as is required for all input files:

```
begin sierra crush_124_spheres
```

We now need to define the functions used with this problem. The boundary conditions require a function for the initial velocity, as follows:



(a) Additional Crush of Spheres

(b) Final Deformed Configuration

Figure A.2: Mesh with blue and green surfaces removed to show internal spheres (yellow) after initial crush shown in Figure A.1 (b).

```
begin function constant_velocity
  type is piecewise linear
  ordinate is velocity
  abscissa is time
  begin values
    0.0 30.0
    1.0 30.0
  end values
end
```

To define the boundary conditions, we need to define the direction for the initial velocity—this is in the y-direction. We could also choose to simply specify the Y component for the initial condition, but this input file uses directions.

```
define direction y_axis with vector 0.0 1.0 0.0
```

Next we define the material models used for this analysis. This problem has two materials: steel for the box, and lead for the spheres. Both materials use the elastic-plastic material model (denoted as `elastic_plastic`).

```
begin material steel
  density = 7871.966988

  begin parameters for model elastic_plastic
    youngs modulus = 1.999479615e+11
    poissons ratio = 0.33333
    yield stress = 275790291.7
    hardening modulus = 275790291.7
    beta = 1.0
  end parameters for model elastic_plastic
end material steel
```

```

begin material lead
  density = 11253.30062

  begin parameters for model elastic_plastic
    youngs modulus = 1.378951459e+10
    poissons ratio = 0.44
    yield stress = 13789514.59
    hardening modulus = 0.0
    beta = 1.0
  end parameters for model elastic_plastic

end material lead

```

Now we define the finite element mesh. This includes specification of the file that contains the mesh, and a list of all the element blocks we will use from the mesh and the material associated with each block. The name of the file is `crush_124_spheres.g`. The specification of the database type is optional—ExodusII is the default. Currently, each element block must be defined individually. For this particular problem, all the spheres are the same element block. Each sphere is a distinct geometry entity, but all spheres constitute one element block in the Exodus II database. The three element blocks that make up the box and lid all reference the same material description. The material description is *not* repeated three times. The material description for steel appears once and is then referenced three times.

```

begin finite element model mesh1
  Database Name = crush_124_spheres.g
  Database Type = exodusII

  begin parameters for block block_1
    material = steel
    model = elastic_plastic
  end parameters for block block_1

  begin parameters for block block_2
    material = steel
    model = elastic_plastic
  end parameters for block block_2

  begin parameters for block block_3
    material = steel
    model = elastic_plastic
  end parameters for block block_3

  begin parameters for block block_4
    material = lead
    model = elastic_plastic
  end parameters for block block_4

```

```
end finite element model mesh1
```

As an alternative to referencing the material description for steel three times as done above, you could define multiple element blocks simultaneously on the same command line. Thus, the three element block specifications with the material `steel` could be consolidated into one, as follows:

```
begin parameters for block block_1 block_2 block_3
  material = steel
  model = elastic_plastic
end parameters for block block_1 block_2 block_3
```

At this point we have finished specifying physics-independent quantities. We now want to set up the Presto procedure and region, along with the time control command block. We start by defining the beginning of the procedure scope, the time control command block, and the beginning of the region scope. Only one time stepping block command block is needed for this analysis. The termination time is set to 7×10^{-4} .

```
begin presto procedure Apst_Procedure

begin time control
  begin time stepping block p1
    start time = 0.0
    begin parameters for presto region presto
      time step scale factor = 1.0
      time step increase factor = 2.0
      step interval = 25
    end parameters for presto region presto
  end time stepping block p1

  termination time = 7.0e-4
end time control

begin presto region presto
```

Next we associate the finite element model we defined above (`mesh1`) with this presto region.

```
use finite element model mesh1
```

Now we define the boundary conditions on the problem. We prescribe the velocity on the top surface of the box (`nodelist_100`) to crush the spheres, and we confine the bottom surface of the box (`nodelist_200`) not to move. Although we use node sets to define these boundary conditions, we could have used the corresponding side sets.

```

begin prescribed velocity
  node set = nodelist_100
  direction = y_axis
  function = constant_velocity
  scale factor = -1.0
end

begin fixed displacement
  node set = nodelist_200
  components = Y
end

```

Now we define the contact for this problem. For this problem, we want all four element blocks to be able to contact each other. In this case, we simply define the same contact characteristics for all interactions. Since no friction model is defined in the block below, the contact defaults to frictionless contact. (Numerous options exist to control the contact algorithm. The options you choose will affect contact algorithm efficiency and solution accuracy. See Chapter 8 to determine how to set input for the `CONTACT DEFINITION` command block to obtain the best level of efficiency and accuracy for your particular problem.)

```

begin contact definition
  skin all blocks = on
  begin interaction defaults
    general contact = on
    self contact = on
  end
end

```

Now we define what variables we want in the results file and how often we want this file to be written. Here we request files written every 7×10^{-6} sec of analysis time. This will result in results output at one hundred time steps (plus the zero time step) since the termination time is set to 7×10^{-4} sec. The output file will be called `crush_124_spheres.e`, and it will be an Exodus II file (the database type command is optional; it defaults to ExodusII). The variables we are requesting are the displacements and external forces at the nodes, the rotated stresses for the elements, the time step increment, and the kinetic energy.

```

begin Results Output output_presto
  Database Name = crush_124_spheres.e
  Database Type = exodusII
  At Time 0.0, Increment = 7.0e-6
  nodal displacement as displ
  nodal force_external as fext
  element stress as stress
  global KineticEnergy as KE
  global timestep
end

```

Now we end the presto region, presto procedure, and sierra blocks to complete the input file.

```
        end presto region presto
    end presto procedure Apst_Procedure
end sierra crush_124_spheres
```

Here is the resulting full input file for this problem:


```

begin sierra crush_124_spheres
  begin function constant_velocity
    type is piecewise linear
    ordinate is velocity
    abscissa is time
    begin values
      0.0 30.0
      1.0 30.0
    end values
  end function constant_velocity
  define direction y_axis with vector 0.0 1.0 0.0

  begin material steel
    density = 7871.966988

    begin parameters for model elastic_plastic
      youngs modulus = 1.999479615e+11
      poissons ratio = 0.33333
      yield stress = 275790291.7
      hardening modulus = 275790291.7
      beta = 1.0
    end parameters for model elastic_plastic

  end material steel

  begin material lead
    density = 11253.30062

    begin parameters for model elastic_plastic
      youngs modulus = 1.378951459e+10
      poissons ratio = 0.44
      yield stress = 13789514.59
      hardening modulus = 0.0
      beta = 1.0
    end parameters for model elastic_plastic

  end material lead

  begin finite element model mesh1
    Database Name = crush_124_spheres.g
    Database Type = exodusII

    begin parameters for block block_1
      material = steel
      model = elastic_plastic
    end parameters for block block_1

```

```

begin parameters for block block_2
  material = steel
  model = elastic_plastic
end parameters for block block_2

begin parameters for block block_3
  material = steel
  model = elastic_plastic
end parameters for block block_3

begin parameters for block block_4
  material = lead
  model = elastic_plastic
end parameters for block block_4

end finite element model mesh1

begin presto procedure Apst_Procedure

begin time control
  begin time stepping block p1
    start time = 0.0
    begin parameters for presto region presto
      time step scale factor = 1.0
      time step increase factor = 2.0
      step interval = 25
    end parameters for presto region presto
  end time stepping block p1

  termination time = 7.0e-4
end time control

begin presto region presto

  use finite element model mesh1

  begin prescribed velocity
    node set = nodelist_100
    direction = y_axis
    function = constant_velocity
    scale factor = -1.0
  end prescribed velocity

  begin fixed displacement
    node set = nodelist_200
    components = Y
  end fixed displacement

```

```

begin contact definition
  skin all blocks = on
  begin interaction defaults
    general contact = on
    self contact = on
  end
end

begin Results Output output_presto
  Database Name = crush_124_spheres.e
  Database Type = exodusII
  At Time 0.0, Increment = 7.0e-6
  nodal displacement as displ
  nodal force_external as fext
  element stress as stress
  global KineticEnergy as KE
  global timestep
end results output output_presto

end presto region presto
end presto procedure Apst_Procedure
end sierra crush_124_spheres

```


Appendix B

Implicit Quasistatic Example Problem

This appendix provides an example problem to illustrate the construction of an input file for an analysis. The problem is modeled after a pencil/eraser that is pressed against and rubbed across a tablet. The pencil, eraser and tablet are represented by blocks 1, 2 and 3 respectively. Block 4 is used to apply kinematics directly to the eraser via tied contact between block 4 and block 2. Block 1 is superfluous except when viewed together with blocks 1, 2 and 3. Then they closely resemble a pencil with an eraser on a tablet. The problem demonstrates the overall structure of an input file and includes the use of both tied and frictional contact simultaneously, the multilevel solver, and a linear solver for preconditioning. A schematic of the problem is shown in Figure B.1 and the mesh is shown in Figure B.2.

The problem kinematics and loading are described briefly here. The loading has two phases: preload and sliding. In the preload phase, the tablet (block 3) is kinematically fixed in the x - y plane and a vertical load (in the z -direction) is applied to the tablet which presses the tablet against the eraser (block 2). The eraser is kinematically fixed to block 4 via tied contact (see Figure B.2). Block 4 is held fixed in all three coordinate directions for the compression phase. The initial configuration and deformations of the preload phase are shown in the first two snapshots on the left in Figure B.1. In the sliding phase, block 4 is kinematically prescribed to move along the

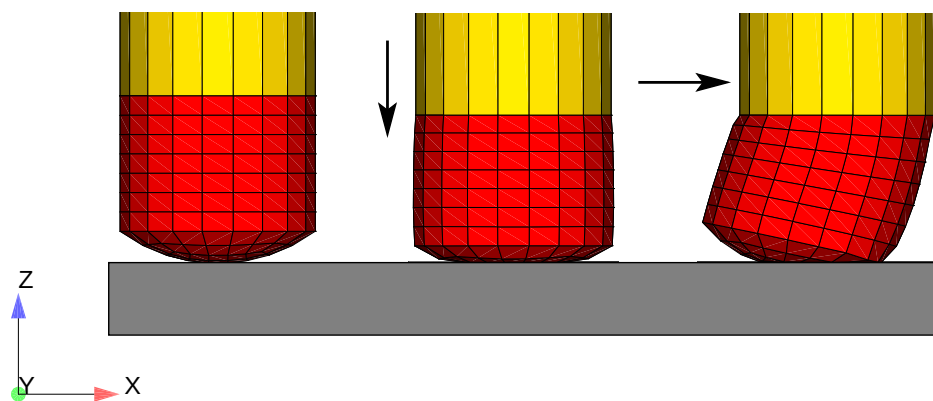


Figure B.1: Eraser schematic; Block 1 (yellow) is pencil, block 2 (red) is eraser, block 3 (gray) is tablet, block 4 (not shown) is tied to eraser and has the kinematics applied to it.

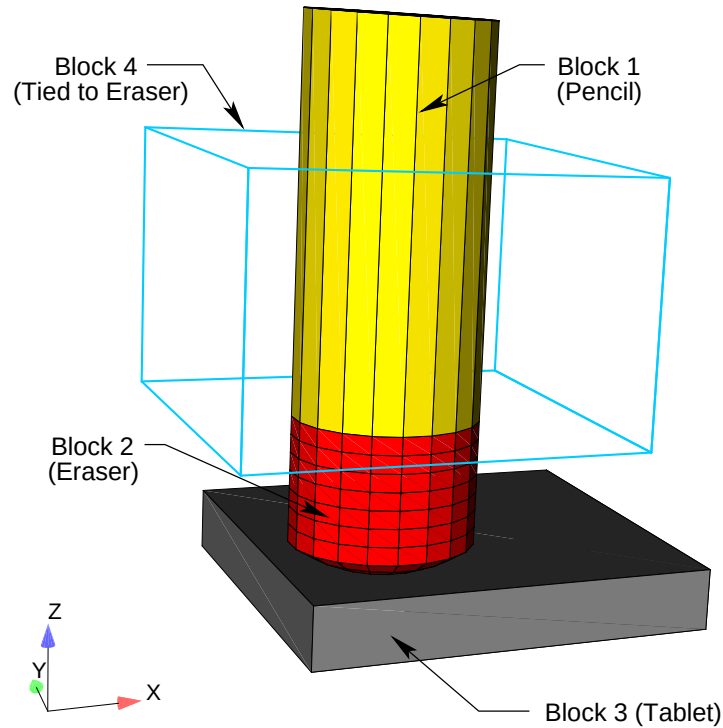


Figure B.2: Complete eraser mesh

x -direction, thus sliding or dragging the eraser along the tablet (see Figure B.1) while the tablet force is held constant.

The input file is described below, with comments to explain every few lines. Following the description, the full input file is listed again. All character strings in the input file are presented in lowercase, which is an acceptable format in Adagio.

The input file starts with a `begin sierra` statement, as is required for all input files:

```
begin sierra eraser
```

We begin by defining vectors corresponding to the coordinate axes. These vectors/directions will be used to define the input for boundary condition blocks that come later.

```
define direction X with vector 1.0 0.0 0.0
define direction Y with vector 0.0 1.0 0.0
define direction Z with vector 0.0 0.0 1.0
```

We now need to define the functions used for this problem. The boundary conditions require a function for the tablet force and the sliding motion. Both the tablet force and sliding motion are prescribed as functions of time.

```

begin function slide
  type is piecewise linear
  begin values
    0.0      0.0
    1.0      0.0
    2.0      1.0
    3.0      1.0
  end values
end function slide

begin function tablet_force
  type is piecewise linear
  begin values
    0.0      0.0
    1.0      1.0
    3.0      1.0
  end values
end function tablet_force

begin function zero
  type is constant
  begin values
    0.0
  end values
end function zero

```

The tablet force ramps up over the time interval (0.0,1.0) and then is held constant over the interval (1.0,3.0). The sliding function is zero over the time interval (0.0,1.0) and then it linearly increases over the interval (1.0,2.0). Next, we define the material properties and models that are used in this problem. In this example, we define two sets of material properties: one is called stiff (pencil, tablet, and block 4), and the second is called soft (eraser). We use a linear elastic constitutive model for both cases.

```

begin material stiff
  density = 1.0
  begin parameters for model elastic
    youngs modulus = 1.e5
    poissons ratio = 0.3
  end parameters for model elastic
end material stiff

begin material soft
  density = 1.0
  begin parameters for model elastic
    youngs modulus = 1000.
    poissons ratio = 0.3
  end parameters for model elastic

```

```
end material soft
```

Now, we define the finite element mesh. This includes specification of the file that contains the mesh, a list of all the element blocks we will use from the mesh and the material associated with each block. The name of the file is `eraser.g`. The specification of the database type is optional-ExodusII is the default. Currently, each element block must be defined individually. The tablet, pencil and block 4 all reference the same material description (stiff). The material description is not repeated three times. The material description for stiff appears once and is then referenced three times.

```
begin finite element model mesh1
  database name = eraser.g
  database type = exodusII
  begin parameters for block block_1 #Pencil
    material = stiff
    model = elastic
  end
  begin parameters for block block_2 #Eraser
    material = soft
    model = elastic
  end
  begin parameters for block block_3 #Tablet
    material = stiff
    model = elastic
  end
  begin parameters for block block_4 #dummy block
    material = stiff
    model = elastic
  end
end
end finite element model mesh1
```

At this point we have finished specifying physics-independent quantities. We now want to set up the Adagio procedure and region, along with the time control command block. We start by defining the beginning of the procedure scope, the time control command block, and the beginning of the region scope. Three time stepping command blocks are used in this analysis. The termination time is set to 1.8. Having multiple time blocks is useful because we can make some solver options a function of the time block.

```
begin adagio procedure agio_procedure
  begin time control
    begin time stepping block preload
      start time = 0.0
      begin parameters for adagio region adagio
        number of time steps = 5
      end parameters for adagio region adagio
    end time stepping block preload
```



```

begin time stepping block slide_1
  start time = 1.0
  begin parameters for adagio region adagio
    number of time steps = 1
  end parameters for adagio region adagio
end time stepping block slide_1
begin time stepping block slide_2
  start time = 1.1
  begin parameters for adagio region adagio
    number of time steps = 10
  end parameters for adagio region adagio
end time stepping block slide_2
termination time = 1.8
end time control

begin adagio region adagio

```

Next we associate the finite element model we defined above (mesh1) with this Adagio region.

```

use finite element model mesh1

```

Now we define the kinematic boundary conditions. First, we prescribe the displacement of surface_200 which is part of block 4. We fix this surface in both the x and z directions and prescribe the horizontal displacement along the X -direction using the previously defined functions `zero` and `slide`.

```

### movement of pencil prescribed by block 4
begin prescribed displacement
  surface = surface_200
  direction = X
  function = slide
  scale factor = 3.0
end prescribed displacement
begin fixed displacement
  surface = surface_200
  components = Y Z
end fixed displacement

```

Next, we fix the tablet and prevent it from moving in the X - Y plane. We do this on all tablet nodes (nodelist_111 and nodelist_112).

```

### Constraints on tablet
begin fixed displacement
  node set = nodelist_111
  components = X Y

```

```

end fixed displacement
begin fixed displacement
  node set = nodelist_112
  components = X Y
end fixed displacement

```

Finally, we prescribe preload force on the tablet which compresses the tablet against the eraser.

```

### Tablet force
begin prescribed force
  node set = nodelist_111
  direction = Z
  function = tablet_force
  scale factor = 100.0
end prescribed force

```

We now define the contact for this problem. Here we need to define tied contact between block 4 and the eraser. In addition, we have frictional sliding contact between the tablet and the eraser.

```

### block 4 tied to eraser ###
begin contact definition
  #
  # define contact surfaces.
  contact surface surf_200 contains surface_200
  contact surface surf_110 contains surface_110
  contact surface surf_11 contains surface_11 # Tablet
  contact surface surf_10 contains surface_10 # Eraser
  #
  begin interaction
    surfaces = surf_200 surf_110
    friction model = tied
  end interaction

  begin interaction
    surfaces = surf_11 surf_10
    friction model = fric
  end interaction

  begin constant friction model fric
    friction coefficient = 0.8
  end

end contact definition

```

Now we define what variables we want in the output file and how often we want the output file to be written. The output file will be called eraser.e, and it will be an ExodusII file (the database type

command is optional; it defaults to ExodusII). The variables we are requesting are the displacements, velocities, and contact diagnostics at the nodes, and stresses and strains at the elements.

```
begin results output output_adagio
  database name = eraser.e
  database type = exodusII
  at step 0, increment = 1
  nodal displacement as displ
  nodal velocity as vel
  nodal contact_tangential_direction as cont_tan
  nodal contact_normal_direction as cont_norm
  nodal contact_accumulated_slip_vector as cont_slip_vec
  nodal contact_status as cont_stat
  nodal contact_normal_traction_magnitude as norm_trac_mag
  nodal contact_tangential_traction_magnitude as tan_trac_mag
  nodal contact_incremental_slip_magnitude as cont_Islip_mag
  nodal contact_accumulated_slip as cont_Aslip
  nodal contact_frictional_energy_density as cont_fric_E
  nodal contact_area
  element stress as stress
  element log_strain as strain
end results output output_adagio
```

The final part of the input deck is the Adagio solver commands. Because we have sliding contact, we must use the solver command block. Nested inside the solver block, we define the control contact block as well as the nonlinear conjugate gradient (CG) solver block.

```
begin solver

  begin loadstep predictor
    type = secant
    slip scale factor = 0.0
  end loadstep predictor

  level 1 predictor = none

  begin control contact name
    target relative residual = 1e-3
    Maximum Iterations      = 200
    maximum slip iterations  = 1
    maximum line search step = 1.00
    minimum line search step = 0.50
    slip nodes               = all_nodal
  end control contact name

  begin cg
    target relative residual = 1.e-4
```

```

        minimum residual improvement = 0.5
        begin full tangent preconditioner
            linear solver = feti
        end full tangent preconditioner
    end cg
end solver

```

After defining the Adagio region and procedure blocks, we close them using the lines:

```

        end adagio region adagio
    end adagio procedure agio_procedure

```

The final detail that we need to define for this file is the linear solver. In the above solver command block, we included the `full tangent preconditioner` block, which has a nested command line `linear solver = feti`. This command line refers to a linear solver called `feti` that must be defined outside the “Procedure” scope but within the “Sierra” scope and can come at the top of the file before the “Procedure” definition or after it as is the case here. The input `feti` on this line command is a user defined string that can have any useful label. The following command block defines the linear solver labeled `feti`. Note that the command block has the label `feti` at the end of the `Begin` line and that this label is referred to from within the above `full tangent preconditioner` command block. The FETI parameters are all set to reasonable values for most problems, so there is no need to set any of them, but if it were necessary to set any of them to non-default values, that would be done within the `feti equation solver` block.

```

    begin feti equation solver feti
        # use default settings
    end feti equation solver feti

```

Finally, we finish the input file and close the sierra command block:

```

end sierra eraser

```

Appendix C

Consistent Units

This appendix describes common consistent sets of units. Maintaining a consistent set of units is crucial in using Sierra/SM when entering material properties and interpreting results. The only variables that have intrinsic units are rotations, which are in radians. All other variables depend on the consistent set of units that the user uses in inputting the material properties and dimensioning the geometry.

A consistent set of units is made by picking the base units. When using the International System (SI), these are length, mass, and time, while in the English unit systems, these base units are length, force, and time. All other units are then derived from these base units. Table C.1 provides several examples of commonly used consistent sets of units. In general, the names of the unit systems in this table are taken from the names of the base units. For example, CGS stands for (centimeters, grams, seconds) and IPS stands for (inches, pounds, seconds).

One of the most common mistakes related to consistent units comes in when entering density. For example, in the IPS system, a common error is to enter the density of stainless steel as $0.289 \text{ lb}/\text{in}^3$, when it should be entered as $7.48\text{e-}4 \text{ lb} \cdot \text{s}^2/\text{in}$. The weight per unit volume should be divided by the gravitational constant ($386.4 \text{ in}/\text{s}^2$ in this case) to obtain a mass per unit volume.

Table C.1: Consistent Unit Sets

Unit	Unit System				
	SI	CGS	IPS	FPS	MMTS
Mass	kg	g	$\frac{lb \cdot s^2}{in}$	$slug$	$tonne$
Length	m	cm	in	ft	mm
Time	s	s	s	s	s
Density	$\frac{kg}{m^3}$	$\frac{g}{cm^3}$	$\frac{lb \cdot s^2}{in^4}$	$\frac{slug}{ft^3}$	$\frac{tonne}{mm^3}$
Force	N	dyn	lb	lb	N
Pressure	Pa	$\frac{dyn}{cm^2}$	psi	psf	MPa
Moment	$N \cdot m$	$dyn \cdot cm$	$in \cdot lb$	$ft \cdot lb$	$N \cdot mm$
Temperature	K	K	$^{\circ}R$	$^{\circ}R$	K
Energy	J	erg	$lb \cdot in$	$lb \cdot ft$	mJ
Velocity	$\frac{m}{s}$	$\frac{cm}{s}$	$\frac{in}{s}$	$\frac{ft}{s}$	$\frac{mm}{s}$
Acceleration	$\frac{m}{s^2}$	$\frac{cm}{s^2}$	$\frac{in}{s^2}$	$\frac{ft}{s^2}$	$\frac{mm}{s^2}$

Appendix D

Constraint Enforcement Hierarchy

When a node has multiple constraints, they are enforced in a specific order. Table D.1 shows the order of enforcement of the various types of constraints.

Table D.1: Constraint Enforcement Order

1	Contact
2	MPC
3	Kinematic
4	Rigid Body

If any of the constraints are in conflict, the last constraint enforced will override previously enforced constraints. For example, if a kinematic boundary condition and a MPC are both active on a node and conflict with each other, the MPC will be enforced first, followed by the kinematic boundary condition. As a result, the kinematic boundary condition will be enforced and will override the MPC.

Additionally, a node may have multiple instances of one type of constraint. For example a node could have multiple conflicting kinematic constraints. For such cases generally the last constraint defined in the input deck will be enforced overriding any conflicting earlier constraints. Only the part of previous constraints orthogonal to the last constraint will be enforced.

Example 1:

```
BEGIN INITIAL VELOCITY
  NODE SET = nodelist_1
  COMPONENT = X
  MAGNITUDE = 0.1
END
BEGIN INITIAL VELOCITY
  NODE SET = nodelist_1
  COMPONENT = Y
```

```
MAGNITUDE = 0.2  
END
```

The two velocity boundary conditions are orthogonal thus combine normally, each node in `nodelist_1` will be given a velocity of (0.1, 0.2, 0.0).

Example 2:

```
DEFINE DIRECTION angle WITH VECTOR 1 1 0  
BEGIN INITIAL VELOCITY  
  NODE SET = nodelist_1  
  DIRECTION = angle  
  MAGNITUDE = 1.0  
END  
BEGIN INITIAL VELOCITY  
  NODE SET = nodelist_1  
  COMPONENT = X  
  MAGNITUDE = 0.5  
END
```

The two velocity boundary conditions are not orthogonal. In evaluating the first boundary condition the nodes will be given a velocity of (0.707107, 0.707107, 0.0). Next the second boundary condition is evaluating overriding the non-orthogonal portion first resulting in a final nodal velocity of (0.5, 0.707107, 0.0).

Appendix E

Improving Explicit Dynamics Performance

In general, the default settings in Sierra/SM explicit dynamics are chosen for robustness and accuracy over speed. When appropriate, defaults are changed in new releases as new options are shown to be sufficiently robust and accurate for a wide range of problems. In this appendix, we describe several approaches that can help to improve the execution speed, though potentially at the expense of robustness or accuracy.

Increase the critical time step

The critical time step is generally controlled by three factors: element size, material stiffness, and material density. Smaller elements drive the critical time step down, as do stiffer materials. Denser materials have the opposite effect of increasing the critical time step.

Improve the mesh quality The critical time step for explicit dynamics is often set by the smallest element. To identify the particular element which is limiting the simulation time step, the critical time step of each element can be output for plotting using the element variable `timestep`, see Table 9.9.

Mass scaling This is a fairly common approach used by many commercial finite element codes to increase the time step. A range of options are provided in Sierra/SM to apply mass scaling dynamically throughout the analysis runtime, see Section 3.3.1. However, mass scaling can affect the solution in critical areas and must be used with caution.

Lanczos time step This is an alternative method for computing the critical time step. The default element-based time step is sometimes more than 30% smaller than the true critical time step, especially if element sizes and material properties vary significantly and suddenly throughout the domain. The Lanczos algorithm computes a scaling factor which can bring the time step closer to the true critical value; this can result in a significant reduction in analysis time. However, for some analyses the Lanczos method does not provide a significantly improved time step, and computing the Lanczos time step estimate is expensive and thus should be done only occasionally during the analysis. And in a few problems, this approach can lead to unstable solutions. See Section 3.2.1.5 for further details.

Node-based time step This is another approach to improve the estimate of the critical time step and often leads to larger time steps than the default element-based time step. Compared to

the Lanczos estimate, the node-based method may still underestimate the critical time step but requires less overhead to compute. See Section 3.2.3.1 for further details.

Improve contact performance

Appropriate contact tolerances It is almost always best (for both performance and robustness) to use the default contact search tolerances. Larger search tolerances will only slow down the simulation.

Mesh rebalancing Use mesh rebalancing to avoid significant loss of performance for problems with large deformations and evolving contact surface interactions see Section 6.8.

Mesh decomposition The initial decomposition of the mesh can have a sizable effect on the parallel performance of a model. For example, a Recursive Coordinate Bisection (RCB) decomposition can sometimes lead to improved load balancing compared to the default decomposition; see also the SEACAS manuals [1, 2, 3].

Self contact off Not using self contact can help decrease the total time needed for tracking and reduce the number of contact interactions to assess. If possible, specify the needed contact interactions without using self contact. See Chapter 8 for more information.

Improve computational efficiency

Reduce the number of side sets, node sets and blocks A large number of side sets, node sets and/or blocks adversely affects the memory layout and can lead to both larger memory usage and longer execution times. Try to define boundary conditions using as few side sets/node sets as possible. Also try to remove node sets, sides sets and blocks which are not explicitly used in the input deck.

Optimal number of mesh partitions If the mesh is partitioned such that there are less than around 1000 elements per processor, the parallel performance may become significantly compromised. Try to aim for 2000-10000 elements per processor. This recommendation may change as computer architectures evolve.

Less frequent checks for element death Element death can lead to significant slowdowns if a large number of criteria are applied every time step. To reduce this effect, assess element death only occasionally, see Section 6.5.3. Also, if possible try to limit the areas that have both element death and contact.

Use the uniform-gradient hex8 element Whenever performance improvements are made to the elements in Sierra/SM they are generally first applied to the uniform-gradient hex8 element. Other element types may also be upgraded or addressed only as needs arise.

Use an alternative material model Some material models have large numbers of function look ups (for instance the orthotropic crush model). These models typically have poor performance. Elastic models are typically significantly faster but may not be appropriate for many analyses.

The Sierra/SM team is interested in assisting with any performance issues. Please contact them if you have questions, example performance timings, or would like help in getting better performance out of a Sierra/SM explicit dynamics simulation.

References

- [1] T.A. Laursen, S.W. Attaway, and R.I. Zadoks. SEACAS theory manuals: Part I. problem formulation in nonlinear solid mechanics. Technical Report SAND98-1760/1, Sandia National Laboratories, Albuquerque, NM, 1998. [pdf](#).
- [2] T.A. Laursen, S.W. Attaway, and R.I. Zadoks. SEACAS theory manuals: Part II. nonlinear continuum mechanics. Technical Report SAND98-1760/2, Sandia National Laboratories, Albuquerque, NM, 1998. [pdf](#).
- [3] T.A. Laursen, S.W. Attaway, and R.I. Zadoks. SEACAS theory manuals: Part III. finite element analysis in nonlinear solid mechanics. Technical Report SAND98-1760/3, Sandia National Laboratories, Albuquerque, NM, 1999. [pdf](#).

Appendix F

Troubleshooting Guide for Implicit Convergence

Implicit simulations often encounter convergence problems that are difficult to diagnose and fix. Below is a list of commonly used techniques for obtaining convergence in implicit problems.

When using a multi-level solver such as control contact, two distinct convergence issues are often encountered: 1) the model problem (i.e., the level 0 solver) fails to converge, and 2) control contact (i.e., the level 1 solver) fails to converge. The following suggestions may help to obtain convergence in either case:

- **Add additional constraints to remove rigid body modes.** If the problem is depending on contact to remove rigid body modes, it can be difficult for the solver to converge due to poor conditioning of the system. If there is a way to remove the rigid body modes without relying on contact then these problems are better posed and easier for the solver to handle.
- **Add tangent diagonal scaling.** If it is not possible to add additional constraints to remove rigid body modes then tangent diagonal scale can avoid some of the ill-conditioning of the problem. Because this option only modifies the preconditioner, it will not affect the final converged solution significantly. However, if too much tangent diagonal scaling is used then the solver convergence rate may slow down. See Section [4.2.2](#) for further discussion.
- **Set a reasonably small number of maximum iterations and a high acceptable residual for the model problem.** This is useful because the contact constraints for the first model problem solve may be over-constrained or otherwise invalid. These inconsistencies are resolved after the first contact iteration, and setting the maximum iterations and a high acceptable residual allows the simulation to get past this hurdle. Adjusting these parameters will not affect the final converged solution since the overall solution will still be checked against the convergence tolerances of the level 1 control contact solve.
- **Try the full tangent preconditioner.** The full tangent preconditioner often makes problems converge in significantly fewer iterations and is typically more efficient than other available preconditioners.
- **Take smaller step sizes.** This can help reduce the amount of nonlinearity introduced in the load step, making the problem easier to solve.
- **Turn off load step predictor.** This can help because the interactions are defined in the initial predicted configuration. Predicting the initial configuration can be very difficult and no prediction is always better than a bad prediction. See Section [4.9](#) for details.
- **Turn off adaptive penalty.** The adaptive penalty algorithm attempts to improve the con-

vergence rate by adjusting the penalty parameters but is sometimes unsuccessful. Without adaptive penalty convergence is typically slower but more robust.

- **Turn off adaptive time stepping.** Adaptive time stepping can mask larger issues by pushing past the real issue and later spawning a fictitious issue.
- **If using frictionless contact, try a frictional model.** Frictional models have tangential stiffness that prevents sliding modes, making the model problem easier to solve.
- **Solve the model problem to a tighter target tolerance.** The penalty kernel for augmented Lagrange comes from the model problem solve. So, if the model problem is not solved to a sufficiently small tolerance then there is little hope that the Lagrange multiplier will converge with an acceptable residual tolerance.
- **Try a different preconditioner.** The full-tangent preconditioner is sensitive to rigid body modes but can handle body forces better than the nodal preconditioner. Alternatively, the nodal preconditioner is insensitive to rigid body modes but is very slow to converge body forces.
- **Use iteration plot to diagnose problems.** This is useful to see if a rigid body mode is present. It can also be helpful to determine where contact is iterating excessively to diagnose problems which would be difficult to address without a visual aid. See Section 4.1.1 for further discussion.
- **Switch contact constraint formulation.** The node-face contact constraint formulation is better able to handle conflicting constraints but may miss contact in corners or introduce force concentrations that make the problem difficult to solve. Alternatively, face-face constraints give smoother forces but can have more difficulty with constraint conflicts.
- **Avoid constraint conflicts with contact.** Kinematic boundary conditions, MPCs and prescribed rigid bodies can all conflict with contact constraint definitions (see Appendix D). When this occurs convergence can be very poor because the penalty algorithm does not see the other constraints and assumes it can freely move the constrained nodes/faces.
- **Use default normal and tangential search tolerance.** Bad search tolerances can cause the solver to miss constraints or attempt to enforce bad constraints. The default values are generally robust.
- **Use block skinning instead of side-set definitions.** Block skinning gives contact search more information to work with when making projections to define constraints. This extra information can help the search cull incorrect contact constraints.
- **Adjust the AL penalty.** If contact constraints appear to be under-enforced, increasing the AL penalty parameter may help. In particular, for cases in which a very stiff material contacts a very soft material, increasing the AL penalty parameter can sometimes improve convergence. Conversely, a high AL penalty parameter may be over enforcing the constraints, resulting in an ill-conditioned system. If this is the case, decreasing the AL penalty may help, see Section 8.6.1.5.
- **Set a Lagrange tolerance.** This is useful if the maximum gap is acceptably small and the residual in the model problem is acceptably small, but the residual imbalance introduced by a small Lagrange multiplier change prevents control contact convergence, see Section 4.5.1.5.
- **Use Control Damped Solve.** This is useful in problems that have rigid body motion before contact is seated, problems where contact causes rebound or bouncing to occur, problems that involve sliding contact, and other difficult implicit problems. Control Damped Solve adds artificial springs to the nodes where the solve difficulty is occurring, and the spring

force is slowly decreased near zero so the springs do not introduce error into the problem. See Section [4.8](#) for more details.

Appendix G

User-Defined Model Initialization

Multiple methods exist to define the initial state of an analysis model. When initializing the state of a new Sierra/SM analysis from a previous analysis, restart is the recommended method (see Section 9.6). However, in some cases a restart initialization is not possible; for instance, if the initial stress/strain state of the model is coming from an external source such as another code, experimental data, or an analytic solution. Another example is when new blocks are added to an existing analysis and need initial stress/strain/material state that is defined in a secondary analysis.

The methodology of manually initializing a model is sometimes called “poor man’s restart.” Generally many node and element fields need to be set to give a fully consistent initial state. The required field set is different for different element types as shown in Table G.1.

Material state needs to be initialized in a manner that is consistent with other element kinematic quantities, as well. Several elastic materials have no state data, but most materials have a state data array at each element. The name of the material state data field is usually defined by `lame_state_{model_name}`, where `model_name` is the name of the material model. For example the name of the state field for the viscoplastic material is `lame_state_viscoplastic` and the name of the state field for the soil foam model is `lame_state_soil_foam`.

Material state variables also often have aliases, which causes some confusion when using them for restart and initialization purposes. When writing out material state in a preload analysis for initialization in a later analysis output, the following syntax is strongly recommended:

```
BEGIN RESULTS OUTPUT
...
ELEMENT VARIABLES = lame_state_viscoplastic(:) as \textbackslash\#
state_viscoplastic
...
END
```

The ‘(:)’ operator will prevent the use of material state alias names; the material state will be output as a single contiguous field which can then be read back in a single operation. An initialization block like the following should then be used to read back in the material state:

```
BEGIN INITIAL CONDITION
INITIALIZE VARIABLE NAME = lame_state_viscoplastic(:)
VARIABLE TYPE = ELEMENT
```

Table G.1: Required Variables For Consistent Model Initialization

Element	Nodes	Formulation	Element Variables	Nodal Variables
Beam	2	Rod	beam_tshear_rs, beam_tshear_rt, beam_t_axis, cross_sectional_area, left_stretch, unrotated_stress	acceleration, displacement, velocity, rotational_acceleration, rotational_displacement, rotational_velocity
		Cylindrical Joint	cylj_rst_theta1, cylj_rst_theta2, hg_resistance, left_stretch, unrotated_stress	displacement, velocity
Shell	3	Orig_Tri_Shell, C0_Tri_Shell	element_thickness, unrotated_stress	acceleration, displacement, velocity, rotational_acceleration, rotational_displacement, rotational_velocity
	4	BL_Shell (Integration rule = Trapezoid)	membranehourglass, shearhourglass, unrotated_stress	acceleration, displacement, velocity, rotational_acceleration, rotational_displacement, rotational_velocity
		BT_Shell (Integration rule = Trapezoid)	hg_resistance_q4, unrotated_stress	acceleration, displacement, velocity, rotational_acceleration, rotational_displacement, rotational_velocity
Hex	8	Uniform Gradient	rotation, left_stretch, hg_resistance, unrotated_stress	displacement, acceleration, velocity
		Fully Integrated	unrotated_stress	displacement, velocity
Tet	4	Uniform Gradient	left_stretch, unrotated_stress	displacement, acceleration, velocity
		Fully Integrated	left_stretch, unrotated_stress	displacement, acceleration, velocity
	10	Composite_Tet	initial_volume, left_stretch, rate_of_deformation, rotation, stress, unrotated_stress	displacement, velocity
Particle	1	SPH	calc_dens, element_density, element_mass, left_stretch, rotation, total_stress, unrotated_stress	coordinates, acceleration, displacement, velocity, rotational_acceleration, rotational_displacement, rotational_velocity, force_internal

```
    READ VARIABLE = state_viscoplastic  
END
```

In addition to element- and material-specific field data, some boundary-specific conditions may have additional initialization requirements.

Appendix H

Other Scripts Maintained by Development Team

In addition to the large Sierra executables the Sierra/SM development team also maintains several pre- and post-processing scripts that can be helpful for analyst use. These scripts are installed with the Sierra tools in the directory `contrib/testTools/adagio/`. Many of the scripts in this directory are used for regression test utilities and not intended for general use. The scripts that are tested for general use are as follows:

`cth_tracers.py` a utility for mapping pressures from a CTH analysis onto a solid mechanics analysis.

`material_fitting.py` a utility to aid fitting of elastic plastic material constants from test data.

`rotateModel.py` a utility to rotate an exodus results file along with all of its data output fields.

`spheregen.py` a utility to convert a solid element model into spheres for use with particle element formulations such as the Reproducing Kernel Particle Method (RKPM).

`wedgeTo3d.py` a utility to translate the mesh and results from an axisymmetric wedge model into a full 3D model.

Appendix I

Known Issues

The following is a list of known issues in Sierra/SM as of the most recent code release.

Section 4.3: Deactivation of element blocks (see Section 6.1.6.9) does not currently work in conjunction with the full tangent preconditioner in Adagio. To use this capability, one of the nodal preconditioners must be used.

Section 6.2.1: For problems with large rotations, the hourglass energies are known to spike exponentially with increases in total rotation. This behavior is observed for both midpoint and strongly objective strain incrementation and for both incremental and total hourglass formulations. The large rotation issue is a known limitation in solid mechanics for all non-hyperelastic material models.

Section 6.2.12: For problems with rotations, the sph algorithm generates rotational spring like forces on the boundary because of known deficiencies in the algorithm in distinguishing between rigid body rotation and deformation. The sph algorithm does not conserve rotational or angular momentum.

Development Manual: Superelements are not compatible with several modeling capabilities. They cannot be used with element death. They cannot be used with node-based, power method, or Lanczos critical time step estimation methods. They are also not compatible with some preconditioners (such as FETI) for implicit solutions.

Section 7.4.2.2: If a prescribed displacement with the `CYLINDRICAL AXIS` option is applied to nodes that fall on the axis, it will have no effect. Separate boundary conditions should be applied to those nodes to fix them in the plane normal to the axis.

Section 7.4.3.2: If a prescribed velocity with the `CYLINDRICAL AXIS` option is applied to nodes that fall on the axis, it will have no effect. Separate boundary conditions should be applied to those nodes to fix them in the plane normal to the axis.

Section 8.9.6: Attempting to use `GLOBAL SEARCH INCREMENT` with a value greater than 1, especially in a problem that contains shells and/or restart, will, in most cases, cause code failure. A `GLOBAL SEARCH INCREMENT` value greater than 1 will, under the best circumstances, give only a marginal improvement in speed.

Section 9.5: User defined variables (see Section 11.2.4) are not currently supported with heartbeat output.

Section 8: Contact of one rigid body against another is currently inexact. This includes contact of actual meshed rigid bodies, analytic surfaces, or incompressible lofted objects (particles, contact through the thickness of shells, or contact through the radius of beams.) As these contact systems have no elastic response, the contact solver may need many iterations to solve these systems or may fail due to an unsolvable system. This could lead to energy conservation issues in the result of contact between these objects.

Section 4.3: Although multiple linear-solver packages are available for use as preconditioners for solving implicit solid mechanics problems, we strongly recommend that the FETI equation solver or plain CG be used in production analyses. The FETI equation solver is actively maintained and tested by the code development team. The `FETI EQUATION SOLVER` command block is documented in Section 4.4. Other solvers may not be thoroughly tested and may have compatibility problems with some code capabilities such as MPCs and contact.

Many of the more advanced constraint types are not fully compatible with one another. Thus if a single node is involved in multiple types of constraints the resultant behavior may be unstable, particularly if the constraints are not fully orthogonal. Currently supported types of constraints, involve kinematic (such as prescribed velocity), rigid body, periodic, MPC, contact, and spot weld. Kinematic constraints are compatible with contact. MPC constraints are somewhat compatible with contact. However, most other combinations have limited compatibility. Thus for example, a node with both a contact constraint and a periodic constraint may lead to model instability.

I.1 Epic Material Model Issues

Epic (MMM) materials will have problems when used with several of the following capabilities.

- Converting MMM elements to SPH particles
- Given a set of MMM SPH particles adding (via particle conversion) or subtracting (via element death) from that set. Changing the contents of the particle set will cause a redistribution of volume among the integration points, not just addition or subtraction of volume at the changed particle.
- Nodal based tetrahedra, and non-local continuum elements have the same non-local issue as SPH particles.
- Any analysis that uses dynamic mesh refinement or dynamic remeshing of any block of MMM material of any topology will hit the issue.
- Mapping of a solution from one mesh to a different mesh will cause a similar problem.

The problem with the above is that many of the epic material models explicitly store the integration point volume as a material state value; the change of this volume is used to compute an incremental strain rate for equation of state calculations. Any method that yields changes in material point volume unassociated with material deformation will cause these models to have severe accuracy and stability problems. If, for instance, on conversion to particles, a new particle returns a volume of 90% of the old element, the material interprets this as a 10% volumetric compression over a single nanoseconds-long time step; the material can view this as a monstrously large shock. This can cause the material point to introduce an immense amount of artificial energy to the system and drive the model immediately unstable.

In order to avoid these problems when converting MMM materials to a particles, convert them to a standard material type (such as elastic plastic). Additionally do not use the MMM materials when using any of the non-local or mesh dynamic methods listed above.

I.2 SPH Particle Method Time Step Stability Issues

Many particle problems that are unstable at the default time step can be made stable if they are run at a manually reduced time step. There are a number of related issues: SPH time step calculation is not that robust to begin with, particularly in high volumetric compression. If over a single time step, two particles can move more than about one half particle radius length, which can cause an instability. Effectively, this will cause the SPH material to invert. This can be an issue with high velocity impact problems. The relative particle velocities are not taken into account in the critical time step calculation, so the code has no knowledge when this may be occurring.

If this problem occurs, it may be fixed by dropping the global time step scaling factor (say from 1.0 to 0.75 to 0.5) and examining if stability can be obtained. Generally if stability cannot be obtained at a factor of 0.5, there is something other than time step driving the instability.

I.3 Particle Creation Issues

If two SPH particles happen to be created extremely close to one another (for example, less than 10“%” of a particle radius), the particles will fly apart violently, adding significant artificial energy to the system. The closer the particles are, the large the energy will be. If two particles are created exactly on top of one another, the error is infinite. This excessively close particle creation can happen with particle conversion or embedded particles. If the source elements for the particles are heavily deformed and in close proximity, new particles can be created close by to existing particles. Additionally contact and/or particle repulsion may be imperfect in high velocity impact cases, allowing bodies to temporarily occupy the same space. If a newly converted particle happens to be placed on top of an existing particle, a major influx of energy may ensue, causing instability.

Possible input deck fixes include

- reducing the time step,
- reducing the potential for contact or particle repulsion errors,
- adding a “failsafe” death criterion to detect and remove particles that have gone unstable (for example, when these types of errors occur, the problem elements often have astronomically high velocities. Establishing a death criterion of velocity that exceeds, for instance, 10 times the initial impact velocity may detect and immediately remove the problem elements, preventing the stability error from propagating to the rest of the analysis.), or
- creating more robustly enforced contact (the use of more contact momentum balance iterations may help prevent materials—and thus newly created particles—from occupying the same space).

I.4 Bad Element Shapes Yielding Invalid Contact Geometry

Elements will continue to return some material response as long as they maintain positive volume. However, long before the point of absolute element inversion, it is possible for the contact geometry

of elements to become corrupted. For example concave and locally inverted elements may have exterior faces that point the wrong way. An incorrectly pointed face can cause major contact errors to occur either adding artificial energy to the system or preventing solver convergence. These types of extreme deformation can be common in high velocity impacts. Use of a `nodal_jacobian_ratio <= 0.0` death criterion can help avoid this issue. For eight-noded hexahedral elements, the contact geometry for an element is guaranteed to be valid at least until this shape metric reduces to zero.

Index

- 2G SCALING
 - in Incompressible Solid material model, [329](#)
- A
 - in Barlat Plasticity material model, [258](#)
 - in Hosford Plasticity material model, [248](#)
 - in Low Density Foam material model, [293](#)
- A0
 - in Soil and Crushable Foam material model, [286](#)
- A1
 - in Soil and Crushable Foam material model, [286](#)
 - in Swanson material model, [337](#)
 - in Viscoelastic Swanson material model, [340](#)
- A2
 - in Soil and Crushable Foam material model, [286](#)
- ABSCISSA
 - in function, [67](#)
- ABSOLUTE SEARCH TOLERANCE
 - in Procedural Transfer, [864](#), [866](#)
- ACCEPTABLE PRESSURE INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE RELATIVE CONTACT RESIDUAL
 - in Control Contact, [167](#)
 - description of, [168](#)
- ACCEPTABLE RELATIVE PRESSURE INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE RELATIVE RESIDUAL
 - in CG, [140](#)
 - description of, [142](#)
 - in Control Contact, [167](#)
 - description of, [168](#)
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE RELATIVE SDEV INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE RELATIVE STRAIN INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE RELATIVE STRESS INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE RESIDUAL
 - in CG, [140](#)
 - description of, [142](#)
 - in Control Contact, [167](#)
 - description of, [168](#)
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE SDEV INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACCEPTABLE STRESS INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- ACTIVE FOR PROCEDURE
 - in Finite Element Model – in Parameters For Block, [363](#)
 - description of, [371](#)
- ACTIVE PERIODS, [93](#)
 - description of, [93](#)
 - in Adaptive Refinement, [476](#)
 - usage in, [478](#)
 - in Adaptive Time Stepping
 - description of, [197](#)
 - in Cavity Expansion, [576](#)
 - in Centripetal Force, [560](#)
 - in Contact Definition
 - use of, [645](#)
 - in Element Death, [450](#)
 - usage in, [463](#)
 - in Fixed Displacement, [497](#)
 - usage in, [498](#)
 - in Fixed Rotation, [514](#)
 - usage in, [515](#)
 - in Fluid Pressure, [570](#)
 - usage in, [572](#)
 - in Gravity, [558](#)
 - in Implicit Dynamics, [199](#)
 - description of, [199](#)
 - in Inertia Relief, [573](#)
 - usage in, [574](#)
 - in Line Weld, [585](#)
 - in Mass Scaling, [128](#)
 - usage in, [130](#)
 - in Periodic, [512](#)
 - in Pore Pressure, [566](#)
 - usage in, [569](#)

- in Prescribed Acceleration, 508
 - usage in, 511
- in Prescribed Displacement, 499
 - usage in, 503
- in Prescribed Force, 545
 - usage in, 549
- in Prescribed Moment, 550
 - usage in, 553
- in Prescribed Rotation, 516
 - usage in, 520
- in Prescribed Rotational Velocity, 520
 - usage in, 524
- in Prescribed Temperature, 561
 - usage in, 564
- in Prescribed Velocity, 503
 - usage in, 507
- in Pressure, 533
 - usage in, 539
- in REFERENCE AXIS ROTATION
 - usage in, 527
- in Silent Boundary, 578
- in Spot Weld, 579
- in Time Step Initialization, 828
 - usage in, 831
- in Traction, 541
 - usage in, 544
- in User Output, 711
 - description of, 726
- in Viscous Damping, 590
 - usage in, 592
- in Volume Repulsion Old
 - usage in, 595
- in Volume Repulsion Old – in BLOCK SET, 594

ADAGIO PROCEDURE, 87

ADAGIO REGION

- in Adagio Procedure, 87
 - description of, 89

ADAPT SHARP ANGLE

- in Adaptive Refinement, 476
 - usage in, 476

ADAPT SIZE RATIO

- in Adaptive Refinement, 476
 - usage in, 476

ADAPT SMOOTH ANGLE

- in Adaptive Refinement, 476
 - usage in, 476

ADAPT TYPE

- in Adaptive Refinement, 476
 - usage in, 476

ADAPTIVE REFINEMENT, 476

- in Remesh, 472

ADAPTIVE STRATEGY

- in Full Tangent Preconditioner, 153
 - description of, 160

ADAPTIVE TIME STEPPING

- in Adagio Region, 193

ADDITIONAL STEPS

- in Heartbeat Output, 734
- in History Output, 728
- in Output Scheduler, 757
- in Restart Data, 745
- in Results Output, 696
 - description of, 707

ADDITIONAL TIMES

- in Heartbeat Output, 734
- in History Output, 728
- in Output Scheduler, 757
- in Restart Data, 745
- in Results Output, 696
 - description of, 707

AGGRESSIVE CONTACT CLEANUP

- in Element Death, 450
- in Element Death
 - description of, 460

AL PENALTY

- in Contact Definition – in Interaction, 637
- in Contact Definition – in Interaction Defaults, 634
 - description of, 637

ALGORITHM DEBUG LEVEL

- in Zoltan Parameters, 470
 - as default, 470

ALIAS

- in Finite Element Model, 358
 - description of, 362

ALLOWABLE MASS INCREASE RATIO

- in Mass Scaling, 128
 - description of, 130

ALPHA

- in Elastic 3D Orthotropic material model, 297
- in Hill Plasticity material model, 252
- in Hyperfoam material model, 295
- in Implicit Dynamics, 199
 - description of, 199

ANGLE_1_ABSCISSA

- in Elastic Orthotropic Fail material model, 307
- in Elastic Orthotropic material model, 303

ANGLE_1_FUNCTION

- in Elastic Orthotropic Fail material model, 307
- in Elastic Orthotropic material model, 303

ANGLE_2_ABSCISSA

- in Elastic Orthotropic Fail material model, 307
- in Elastic Orthotropic material model, 303

ANGLE_2_FUNCTION

- in Elastic Orthotropic Fail material model, 307
- in Elastic Orthotropic material model, 303

ANGLE_3_ABSCISSA

- in Elastic Orthotropic Fail material model, 307
- in Elastic Orthotropic material model, 303

- ANGLE_3_FUNCTION
 - in Elastic Orthotropic Fail material model, 307
 - in Elastic Orthotropic material model, 303
- ANGULAR VELOCITY
 - in Initial Velocity, 529
 - description of, 531
 - in Rigid Body command block, 442
- AREA
 - in Damper Section, 414
 - in Truss Section, 411
- AT DISCONTINUITY EVALUATE TO
 - in function, 67
- AT STEP
 - in Heartbeat Output, 734
 - in History Output, 728
 - in Output Scheduler, 757
 - in Restart Data, 745
 - in Results Output, 696
 - description of, 707
- AT TIME
 - in Heartbeat Output, 734
 - in History Output, 728
 - in Output Scheduler, 757
 - in Restart Data, 745
 - in Results Output, 696
 - description of, 707
- aupst_check_elem_var, 806
- aupst_check_global_var, 816
- aupst_check_node_var, 806
- aupst_cyl_transform, 855
- aupst_evaluate_function, 805
- aupst_get_elem_centroid, 823
- aupst_get_elem_nodes, 818
- aupst_get_elem_topology, 818
- aupst_get_elem_var, 806
- aupst_get_elem_var_offset, 806
- aupst_get_face_nodes, 818
- aupst_get_face_topology, 818
- aupst_get_global_var, 816
- aupst_get_integer_param, 800
- aupst_get_node_var, 806
- aupst_get_point, 823
- aupst_get_proc_num, 823
- aupst_get_real_param, 800
- aupst_get_string_param, 800
- aupst_get_time, 805
- aupst_put_elem_var, 806
- aupst_put_elem_var_offset, 806
- aupst_put_global_var, 816
- aupst_put_node_var, 806
- aupst_rec_transform, 857
- AXIS OFFSET
 - in Beam Section, 401
- AXIS OFFSET GLOBAL
 - in Beam Section, 401
- AXIS OFFSET VARIABLE
 - in Beam Section, 401
- B
 - in Low Density Foam material model, 293
- B1
 - in Swanson material model, 337
 - in Viscoelastic Swanson material model, 340
- BACK STRESS DYNAMIC RECOVERY CONSTANT
 - in BCJ_MEM material model, 271
- BACK STRESS DYNAMIC RECOVERY TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, 271
- BACK STRESS HARDENING CONSTANT
 - in BCJ_MEM material model, 271
- BACK STRESS HARDENING TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, 271
- BALANCE PROBE
 - in CG, 140
 - description of, 145
 - in Full Tangent Preconditioner, 153
 - description of, 155
- BEAM SECTION, 401
- BENDING HOURGLASS STIFFNESS
 - in Finite Element Model – in Parameters For Block, 363
 - description of, 367
- BENDING HOURGLASS VISCOSITY
 - in Finite Element Model – in Parameters For Block, 363
 - description of, 367
- BETA
 - in Elastic-Plastic material model, 224
 - in Foam Plasticity material model, 289
 - in Implicit Dynamics, 199
 - description of, 199
 - in Multilinear Elastic-Plastic Hardening Model material model, 234
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, 238
 - in Thermoelastic Plastic material model, 244
- BETA METHOD
 - in CG, 140
 - description of, 150
- BIOT'S COEFFICIENT
 - about, 207
- BLOCK
 - description of, 482
 - in Adaptive Refinement, 476
 - usage in, 477
 - in Centripetal Force, 560
 - in Contact Definition – in Contact Surface (block)

- usage in, [612](#)
- in Element Death, [450](#)
 - description of, [452](#)
- in Fixed Displacement, [497](#)
 - usage in, [498](#)
- in Fixed Rotation, [514](#)
 - usage in, [515](#)
- in Gravity, [558](#)
 - usage in, [558](#)
- in Inertia Relief, [573](#)
 - usage in, [573](#)
- in Initial Condition, [489](#)
 - usage in, [490](#)
- in Initial Velocity, [529](#)
 - usage in, [530](#)
- in Line Weld, [585](#)
- in Mass Properties, [448](#)
 - usage in, [448](#)
- in Mass Scaling, [128](#)
 - usage in, [129](#)
- in Pore Pressure, [566](#)
 - usage in, [567](#)
- in Prescribed Acceleration, [508](#)
 - usage in, [509](#)
- in Prescribed Displacement, [499](#)
 - usage in, [500](#)
- in Prescribed Force, [545](#)
 - usage in, [547](#)
- in Prescribed Moment, [550](#)
 - usage in, [551](#)
- in Prescribed Rotation, [516](#)
 - usage in, [518](#)
- in Prescribed Rotational Velocity, [520](#)
 - usage in, [522](#)
- in Prescribed Temperature, [561](#)
 - usage in, [562](#)
- in Prescribed Velocity, [503](#)
 - usage in, [505](#)
- in Procedural Transfer, [864](#)
 - description of, [865](#)
- in Reference Axis, [524](#)
- in Reference Axis Rotation
 - usage in, [526](#)
- in Remesh Block Set, [475](#)
- in Time Step Initialization, [828](#)
 - usage in, [829](#)
- in User Output, [711](#)
 - usage in, [715](#)
- in Viscous Damping, [590](#)
 - usage in, [591](#)
- in Volume Repulsion Old
 - usage in, [595](#)
- in Volume Repulsion Old – in BLOCK SET, [594](#)

BLOCK BY BLOCK

- in Procedural Transfer, [864](#)
 - description of, [866](#)

BLOCK SET

- in Volume Repulsion Old, [594](#)

BOND CUTTING BLOCK

- in Peridynamics Section, [421](#)

BOND DAMAGE MODEL

- in Peridynamics Section, [421](#)

BOND VISUALIZATION

- in Peridynamics Section, [421](#)

Bond-Based Microelastic Peridynamic Material Model

- in Peridynamics, [789](#)

Bond-Based Microplastic Peridynamic Material Model

- in Peridynamics, [789](#)

BULK FUNCTION

- in Mooney-Rivlin material model, [332](#)

BULK MODULUS

- in Barlat Plasticity material model, [258](#)
- in BCJ material model, [268](#)
- in Ductile Fracture material model, [231](#)
- in Elastic 3D Orthotropic material model, [297](#)
- in Elastic Fracture material model, [222](#)
- in Elastic material model, [217](#)
- in Elastic Orthotropic Fail material model, [307](#)
- in Elastic Orthotropic material model, [303](#)
- in Elastic-Plastic material model, [224](#)
- in Elastic-Plastic Power-Law Hardening material model, [228](#)
- in Foam Plasticity material model, [289](#)
- in Hill Plasticity material model, [252](#)
- in Hosford Plasticity material model, [248](#)
- in Hyperfoam material model, [295](#)
- in Incompressible Solid material model, [329](#)
- in Johnson–Cook material model, [265](#)
- in Low Density Foam material model, [293](#)
- in Mooney-Rivlin material model, [332](#)
- in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
- in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
- in Neo Hookean material model, [220](#)
- in Orthotropic Crush material model, [317](#)
- in Orthotropic Rate material model, [321](#)
- in Plane Stress Rate Plasticity material model, [324](#)
- in Power Law Creep material model, [279](#)
- in Soil and Crushable Foam material model, [286](#)
- in Stiff Elastic material model, [335](#)
- in Swanson material model, [337](#)
- in Thermoelastic material model, [219](#)
- in Thermoelastic Plastic material model, [244](#)
- in Viscoelastic Swanson material model, [340](#)
- in Viscoplastic material model, [281](#)
- in Wire Mesh material model, [315](#)

C
in Low Density Foam material model, 293

C01
in Mooney-Rivlin material model, 332

C01 FUNCTION
in Mooney-Rivlin material model, 332

C1
in BCJ material model, 268
in Swanson material model, 337
in Viscoelastic Swanson material model, 340

C10
in BCJ material model, 268
in Mooney-Rivlin material model, 332

C10 FUNCTION
in Mooney-Rivlin material model, 332

C11
in BCJ material model, 268

C12
in BCJ material model, 268

C13
in BCJ material model, 268

C14
in BCJ material model, 268

C15
in BCJ material model, 268

C16
in BCJ material model, 268

C17
in BCJ material model, 268

C18
in BCJ material model, 268

C19
in BCJ material model, 268

C2
in BCJ material model, 268

C20
in BCJ material model, 268

C3
in BCJ material model, 268

C4
in BCJ material model, 268

C5
in BCJ material model, 268

C6
in BCJ material model, 268

C7
in BCJ material model, 268

C8
in BCJ material model, 268

C9
in BCJ material model, 268

CAPTURE TOLERANCE
in Contact Definition – in Interaction, 637
description of, 641

CAVITY EXPANSION, 576

CENTRIPETAL FORCE, 560

CG, 140
in Adagio Region, 140
in Solver, 135

CHARACTERISTIC_LENGTH
in Elastic Orthotropic Fail material model, 307

CHECK GEOMETRY
in Zoltan Parameters, 470
as default, 470

CHECK STEP INTERVAL
in Element Death, 450
description of, 457

CHECK TIME INTERVAL
in Element Death, 450
description of, 457

Classical Material Models
with Peridynamics, 789

COARSE SOLVER
in FETI Equation Solver, 162
description of, 164

COHESIVE MATERIAL
in Element Death, 450

COHESIVE MODEL
in Element Death, 450

COHESIVE SECTION, 384
in Element Death, 450

COHESIVE ZONE INITIALIZATION METHOD
in Element Death, 450

COHESIVE ZONE MODEL
in Contact Definition
description of, 626

COMMUNICATION RATIO THRESHOLD
in Rebalance, 468
description of, 469

COMPONENT
in Fixed Displacement, 497
description of, 498, 515
in Fixed Rotation, 514
description of, 515
in Inertia Relief, 573
in Initial Velocity, 529
description of, 530
in Periodic, 512
in Prescribed Acceleration, 508
description of, 510
in Prescribed Displacement, 499
description of, 501
in Prescribed Force, 545
description of, 547
in Prescribed Moment, 550
description of, 552
in Prescribed Rotational Velocity, 520
description of, 522

- in Prescribed Velocity, 503
- description of, 505
- COMPONENT SEPARATOR CHARACTER
 - in Finite Element Model, 358
 - description of, 363
 - in Restart Data, 745
 - description of, 753
 - in Results Output, 696
 - description of, 706
- COMPONENTS
 - in Fixed Displacement, 497
 - description of, 498, 515
 - in Fixed Rotation, 514
 - description of, 515
 - in MPC, 598
 - in Prescribed Acceleration, 508
 - in Prescribed Displacement, 499
 - in Prescribed Velocity, 503
 - in TIED MPC, 595
- COMPRESSION_COUPLING_FACTOR_11
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSION_COUPLING_FACTOR_22
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSION_COUPLING_FACTOR_33
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_DAMAGE_MODULUS_11
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_DAMAGE_MODULUS_22
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_DAMAGE_MODULUS_33
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_FIBER_STRENGTH_11
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_FIBER_STRENGTH_22
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_FIBER_STRENGTH_33
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_FRACTURE_ENERGY_11
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_FRACTURE_ENERGY_22
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_FRACTURE_ENERGY_33
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_MATRIX_STRENGTH_11
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_MATRIX_STRENGTH_22
 - in Elastic Orthotropic Fail material model, 307
- COMPRESSIVE_MATRIX_STRENGTH_33
 - in Elastic Orthotropic Fail material model, 307
- COMPUTE AT EVERY STEP, 711
 - in User Output
 - description of, 726
- COMPUTE AT HEARTBEAT OUTPUT STEPS, 711
- COMPUTE AT HISTORY OUTPUT STEPS, 711
- COMPUTE AT RESULTS OUTPUT STEPS, 711
- COMPUTE ELEMENT
 - in User Output, 711
- COMPUTE GLOBAL
 - in User Output, 711
- COMPUTE NODAL
 - in User Output, 711
- CONDITIONING
 - in Full Tangent Preconditioner, 153
 - description of, 155
- CONNECTION BLOCK
 - in RBE3, 601
- CONNECTION NODE SET
 - in RBE3, 601
- CONNECTION NODES
 - in RBE3, 601
- CONNECTION SURFACE
 - in RBE3, 601
- CONSTANT FRICTION MODEL
 - in Contact Definition
 - description of, 623
- CONSTANT SPHERE RADIUS
 - in Particle Section, 417
- CONSTRAINT ENFORCEMENT
 - in Full Tangent Preconditioner, 153
 - description of, 155
- CONSTRAINT FORMULATION
 - in Contact Definition – in Interaction, 637
 - in Contact Definition – in Interaction Defaults, 634
 - description of, 636
- Contact
 - with Peridynamics, 792
- CONTACT CLEANUP
 - in Remesh, 472
 - usage in, 473
- CONTACT NODE SET
 - in Contact Definition
 - description of, 612
- CONTACT SURFACE
 - in Contact Definition
 - description of, 611
- CONTACT SURFACE (block)
 - in Contact Definition
 - description of, 612
- CONTACT TIMESTEP
 - in Implicit Dynamics, 199
 - description of, 200
- CONTACT VARIABLES
 - in Contact Definition
 - description of, 663
- CONTROL CONTACT
 - in Solver, 135
 - description of, 167
- CONTROL DAMPED SOLVE

- in Solver, [135](#)
 - description of, [184](#)
- CONTROL FAILURE
 - in Solver, [135](#)
 - description of, [181](#)
- CONTROL STIFFNESS
 - in Solver, [135](#)
 - description of, [177](#)
- COORDINATE SYSTEM, [84](#)
 - in Solid Section, [378](#)
 - in Elastic 3D Orthotropic material model, [297](#)
 - in Elastic Orthotropic Fail material model, [307](#)
 - in Elastic Orthotropic material model, [303](#)
 - in Hill Plasticity material model, [252](#)
- COPY ELEMENT VARIABLE
 - in User Output, [711](#)
 - description of, [722](#)
- COPY VARIABLE
 - in Initial Condition, [489](#)
 - in Pore Pressure, [566](#)
 - description of, [568](#)
 - in Prescribed Acceleration, [508](#)
 - description of, [511](#)
 - in Prescribed Displacement, [499](#)
 - description of, [503](#)
 - in Prescribed Rotation, [516](#)
 - description of, [519](#)
 - in Prescribed Rotational Velocity, [520](#)
 - description of, [524](#)
 - in Prescribed Temperature, [561](#)
 - description of, [563](#)
 - in Prescribed Velocity, [503](#)
 - description of, [507](#)
 - in Pressure, [533](#)
 - description of, [537](#)
- copy_data, [858](#)
- CP12
 - in Barlat Plasticity material model, [258](#)
- CP13
 - in Barlat Plasticity material model, [258](#)
- CP21
 - in Barlat Plasticity material model, [258](#)
- CP23
 - in Barlat Plasticity material model, [258](#)
- CP31
 - in Barlat Plasticity material model, [258](#)
- CP32
 - in Barlat Plasticity material model, [258](#)
- CP44
 - in Barlat Plasticity material model, [258](#)
- CP55
 - in Barlat Plasticity material model, [258](#)
- CP66
 - in Barlat Plasticity material model, [258](#)
- CPP12
 - in Barlat Plasticity material model, [258](#)
- CPP13
 - in Barlat Plasticity material model, [258](#)
- CPP21
 - in Barlat Plasticity material model, [258](#)
- CPP23
 - in Barlat Plasticity material model, [258](#)
- CPP31
 - in Barlat Plasticity material model, [258](#)
- CPP32
 - in Barlat Plasticity material model, [258](#)
- CPP44
 - in Barlat Plasticity material model, [258](#)
- CPP55
 - in Barlat Plasticity material model, [258](#)
- CPP66
 - in Barlat Plasticity material model, [258](#)
- CREATE VISUALIZATION ELEMENTS
 - in Spot Weld, [579](#)
- CREEP CONSTANT
 - in Power Law Creep material model, [279](#)
 - in Viscoplastic material model, [281](#)
- CREEP EXPONENT
 - in Power Law Creep material model, [279](#)
 - in Viscoplastic material model, [281](#)
- CRITERION IS
 - in Element Death – for Always True, [450](#)
 - description of, [456](#)
 - in Element Death – for Element Value Of, [450](#)
 - description of, [454](#)
 - in Element Death – for Global Value Of, [450](#)
 - description of, [455](#)
 - in Element Death – for Nodal Value Of, [450](#)
 - description of, [453](#)
- CRITICAL BIAXIALITY RATIO
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
- CRITICAL CRACK OPENING STRAIN
 - in Ductile Fracture material model, [231](#)
 - in Elastic Fracture material model, [222](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
- CRITICAL NORMAL GAP
 - in Contact Definition – in Cohesive Zone Model
 - description of, [626](#)
- CRITICAL TANGENTIAL GAP
 - in Contact Definition – in Cohesive Zone Model
 - description of, [626](#)
- CRITICAL TEARING PARAMETER
 - in Ductile Fracture material model, [231](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
- CRUSH XX

- in Orthotropic Crush material model, [317](#)
- CRUSH XY
 - in Orthotropic Crush material model, [317](#)
- CRUSH YY
 - in Orthotropic Crush material model, [317](#)
- CRUSH YZ
 - in Orthotropic Crush material model, [317](#)
- CRUSH ZX
 - in Orthotropic Crush material model, [317](#)
- CRUSH ZZ
 - in Orthotropic Crush material model, [317](#)
- CUT OFF STRAIN
 - in Swanson material model, [337](#)
 - in Viscoelastic Swanson material model, [340](#)
- CUTBACK FACTOR
 - in Adaptive Time Stepping
 - description of, [196](#)
- CYCLE COUNT
 - in Restart Data, [745](#)
 - description of, [754](#)
- CYLINDRICAL AXIS
 - in Centripetal Force, [560](#)
 - in Initial Velocity, [529](#)
 - description of, [531](#)
 - in Prescribed Displacement, [499](#)
 - description of, [501](#)
 - in Prescribed Velocity, [503](#)
 - description of, [505](#)
 - in Rigid Body command block, [442](#)
- DAMAGE EXPONENT
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- DAMPER SECTION, [414](#)
- DATABASE NAME
 - in Finite Element Model, [358](#)
 - description of, [361](#)
 - in History Output, [728](#)
 - in Restart Data, [745](#)
 - description of, [747](#)
 - in Results Output, [696](#)
- DATABASE TYPE
 - in Finite Element Model, [358](#)
 - description of, [361](#)
 - in History Output, [728](#)
 - in Restart Data, [745](#)
 - description of, [747](#)
 - in Results Output, [696](#)
- DEATH METHOD
 - in Element Death, [450](#)
 - in Element Death
 - description of, [460](#)
- DEATH ON ILL DEFINED CONTACT
 - in Element Death
 - description of, [459](#)
- DEATH ON INVERSION
 - in Element Death, [450](#)
 - in Element Death
 - description of, [458](#)
- DEATH PROXIMITY
 - in Element Death, [450](#)
 - description of, [461](#)
- DEATH START TIME
 - in Element Death, [450](#)
 - description of, [457](#)
- DEATH STEPS
 - in Element Death, [450](#)
 - description of, [459](#)
- DEBUG ITERATION PLOT
 - in Contact Definition – in Remove Initial Overlap
 - description of, [651](#)
- DEBUG MEMORY
 - in Zoltan Parameters, [470](#)
- DEBUG OUTPUT LEVEL
 - in Remesh, [472](#)
 - usage in, [473](#)
- DEBUG PROCESSOR NUMBER
 - in Zoltan Parameters, [470](#)
- DECAY COEFFICIENT
 - in Plane Stress Rate Plasticity material model, [324](#)
- DEFINE AXIS
 - with point and direction, [79](#)
 - with two points, [79](#)
- DEFINE DIRECTION, [79](#)
- DEFINE POINT, [50](#), [79](#)
- DENSITY
 - in Fluid Pressure, [570](#)
 - in Fluid Pressure
 - usage in, [571](#)
- DENSITY FOR PLASTIC DISSIPATION CALCULATIONS
 - in BCJ_MEM material model, [271](#)
- DENSITY FORMULATION
 - in Particle Section, [417](#)
- DENSITY FUNCTION
 - in Fluid Pressure, [570](#)
 - usage in, [571](#)
- DEPTH
 - in Fluid Pressure, [570](#)
 - in Fluid Pressure
 - usage in, [571](#)
- DEPTH FUNCTION
 - in Fluid Pressure, [570](#)
 - usage in, [571](#)
- DERIVED LOG STRAIN OUTPUT, [467](#)
- DERIVED OUTPUT, [467](#)
- DERIVED STRAIN OUTPUT, [467](#)
- DETECT BREACH, [785](#)

DETERMINISTIC DECOMPOSITION
 in Zoltan Parameters, [470](#)
 DEVIATORIC PARAMETER
 in Membrane Section, [398](#)
 in Solid Section, [378](#)
 DIFFERENTIATE EXPRESSION
 in Function, [67](#)
 DIRECTION
 in Gravity, [558](#)
 in Initial Velocity, [529](#)
 description of, [530](#)
 in Prescribed Acceleration, [508](#)
 description of, [510](#)
 in Prescribed Displacement, [499](#)
 description of, [501](#)
 in Prescribed Force, [545](#)
 description of, [547](#)
 in Prescribed Moment, [550](#)
 description of, [552](#)
 in Prescribed Rotation, [516](#)
 description of, [518](#)
 in Prescribed Rotational Velocity, [520](#)
 description of, [522](#)
 in Prescribed Velocity, [503](#)
 description of, [505](#)
 in Rigid Body command block, [442](#)
 in Traction, [541](#)
 description of, [542](#)
 DIRECTION FOR ROTATION
 in Elastic 3D Orthotropic material model, [297](#)
 in Hill Plasticity material model, [252](#)
 DISTANCE TYPE
 in Initial Condition, [489](#)
 DISTRIBUTE REFERENCE NODE MASS
 in RBE3, [601](#)
 DRILLING STIFFNESS FACTOR
 in Shell Section, [388](#)

 E11
 in Elastic Orthotropic Fail material model, [307](#)
 in Elastic Orthotropic material model, [303](#)
 E11_FUNCTION
 in Elastic Orthotropic Fail material model, [307](#)
 E22
 in Elastic Orthotropic Fail material model, [307](#)
 in Elastic Orthotropic material model, [303](#)
 E22_FUNCTION
 in Elastic Orthotropic Fail material model, [307](#)
 E33
 in Elastic Orthotropic Fail material model, [307](#)
 in Elastic Orthotropic material model, [303](#)
 E33_FUNCTION
 in Elastic Orthotropic Fail material model, [307](#)
 EDGE
 in Heartbeat Output, [734](#)
 in Heartbeat Output – for mesh entities variables
 description of, [737](#)
 in Heartbeat Output – for nearest point variables
 description of, [738](#)
 in History Output, [728](#)
 about, [729](#), [736](#)
 in History Output – for mesh entities variables
 description of, [730](#)
 in History Output – for nearest point variables
 description of, [731](#)
 in Results Output, [696](#)
 EFFECTIVE MODULI MODEL
 in Finite Element Model – in Parameters For Block,
 [363](#)
 description of, [369](#)
 EIGENVALUE CONVERGENCE TOLERANCE
 in Lanczos Parameters, [116](#)
 in Power Method Parameters, [123](#)
 ELASTIC_RATE_COEFFICIENT_11
 in Elastic Orthotropic Fail material model, [307](#)
 ELASTIC_RATE_COEFFICIENT_12
 in Elastic Orthotropic Fail material model, [307](#)
 ELASTIC_RATE_COEFFICIENT_13
 in Elastic Orthotropic Fail material model, [307](#)
 ELASTIC_RATE_COEFFICIENT_22
 in Elastic Orthotropic Fail material model, [307](#)
 ELASTIC_RATE_COEFFICIENT_23
 in Elastic Orthotropic Fail material model, [307](#)
 ELASTIC_RATE_COEFFICIENT_33
 in Elastic Orthotropic Fail material model, [307](#)
 ELEMENT
 in Heartbeat Output, [734](#)
 in Heartbeat Output – for mesh entities variables
 description of, [737](#)
 in Heartbeat Output – for nearest point variables
 description of, [738](#)
 in History Output, [728](#)
 about, [729](#), [736](#)
 in History Output – for mesh entities variables
 description of, [730](#)
 in History Output – for nearest point variables
 description of, [731](#)
 in Inertia Relief, [573](#)
 usage in, [573](#)
 ELEMENT BLOCK SUBROUTINE
 as user subroutine command line, [826](#)
 description of, [827](#)
 in Element Death, [450](#)
 description of, [457](#)
 in Initial Condition, [489](#)
 description of, [494](#)
 in Pore Pressure, [566](#)
 in Time Step Initialization, [828](#)

- description of, [830](#)
 - in User Output, [711](#)
 - description of, [721](#)
- ELEMENT DEATH, [450](#)
 - example of, [464](#)
- ELEMENT GROUPING TYPE
 - in Rebalance, [468](#)
- ELEMENT LIE VARIABLES
 - in Procedural Transfer, [864](#)
 - description of, [865](#)
- ELEMENT NUMERICAL FORMULATION
 - in Finite Element Model – in Parameters For Block, [363](#)
 - description of, [370](#)
- ELEMENT REPRESENTATION
 - in Volume Repulsion Old
 - usage in, [595](#)
 - in Volume Repulsion Old – in BLOCK SET, [594](#)
- ELEMENT VARIABLES
 - in Procedural Transfer, [864](#)
 - description of, [865](#)
 - in Results Output, [696](#)
 - description of, [703](#)
- EMBEDDED BLOCKS
 - in Submodel
 - usage in, [600](#)
- ENCLOSING BLOCKS
 - in Submodel
 - usage in, [600](#)
- ENFORCEMENT
 - in Contact Definition
 - use of, [650](#)
- ENFORCEMENT OPTIONS
 - in Contact Definition
 - description of Options, [660](#)
- EVALUATE EXPRESSION
 - in Function, [67](#)
 - examples of, [71](#)
 - rules and options for composing, [71](#)
- EX
 - in Orthotropic Crush material model, [317](#)
- EXCLUDE
 - in Results Output, [696](#), [705](#)
- EXPANSION RADIUS
 - in Cavity Expansion, [576](#)
- EXTERNAL FORCE CONTRIBUTION OUTPUT
 - NAME, [533](#)
 - in Inertia Relief
 - description of, [574](#)
 - in Pressure
 - description of, [538](#)
- EXTRAPOLATE ELEMENT VARIABLE
 - in User Output, [711](#)
 - description of, [722](#)
- EY
 - in Orthotropic Crush material model, [317](#)
- EZ
 - in Orthotropic Crush material model, [317](#)
- FACE
 - in Heartbeat Output, [734](#)
 - in Heartbeat Output – for mesh entities variables
 - description of, [737](#)
 - in Heartbeat Output – for nearest point variables
 - description of, [738](#)
 - in History Output, [728](#)
 - about, [729](#), [736](#)
 - in History Output – for mesh entities variables
 - description of, [730](#)
 - in History Output – for nearest point variables
 - description of, [731](#)
- FACE VARIABLES
 - in Results Output, [696](#)
 - description of, [701](#)
- FACTOR
 - in BCJ material model, [268](#)
- FAILURE DECAY CYCLES
 - in Line Weld, [585](#)
 - in Spot Weld, [579](#)
- FAILURE ENVELOPE EXPONENT
 - in Line Weld, [585](#)
 - in Spot Weld, [579](#)
- FAILURE FUNCTION
 - in Spot Weld, [579](#)
- FETI EQUATION SOLVER, [162](#)
- FIBER_STRENGTH_RATE_COEFFICIENT_11
 - in Elastic Orthotropic Fail material model, [307](#)
- FIBER_STRENGTH_RATE_COEFFICIENT_12
 - in Elastic Orthotropic Fail material model, [307](#)
- FIBER_STRENGTH_RATE_COEFFICIENT_13
 - in Elastic Orthotropic Fail material model, [307](#)
- FIBER_STRENGTH_RATE_COEFFICIENT_22
 - in Elastic Orthotropic Fail material model, [307](#)
- FIBER_STRENGTH_RATE_COEFFICIENT_23
 - in Elastic Orthotropic Fail material model, [307](#)
- FIBER_STRENGTH_RATE_COEFFICIENT_33
 - in Elastic Orthotropic Fail material model, [307](#)
- FIELD VARIABLE
 - in Pressure, [533](#)
 - description of, [537](#)
- FINAL RADIUS MULTIPLICATION FACTOR
 - in Particle Section, [417](#)
- FINITE ELEMENT MODEL, [358](#)
- FIXED DISPLACEMENT, [497](#)
- FIXED ROTATION, [514](#)
- FLANGE THICKNESS
 - in Beam Section, [401](#)
- FLOW RULE COEFFICIENT CONSTANT

- in BCJ_MEM material model, [271](#)
- FLOW RULE COEFFICIENT TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, [271](#)
- FLOW RULE EXPONENT CONSTANT
 - in BCJ_MEM material model, [271](#)
- FLOW RULE EXPONENT TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, [271](#)
- FLUID PRESSURE, [570](#)
- FLUID SURFACE NORMAL
 - in Fluid Pressure, [570](#)
 - in Fluid Pressure
 - usage in, [571](#)
- FORCE SCALE FACTOR
 - in Centripetal Force, [560](#)
- FORCE STRAIN FUNCTION
 - in Spring Section, [412](#)
- FORCE VALID ACME CONNECTIVITY
 - in Element Death, [450](#)
 - description of, [460](#)
- FORMAT
 - in Heartbeat Output, [734](#)
- FORMULATION
 - in Membrane Section, [398](#)
 - in Particle Section, [417](#)
 - in Solid Section, [378](#)
- Fracture
 - with Peridynamics, [791](#)
- FRACTURE CURVES
 - in Plane Stress Rate Plasticity material model, [324](#)
- FRACTURE STRAIN RATES
 - in Plane Stress Rate Plasticity material model, [324](#)
- FREE SURFACE
 - in Cavity Expansion, [576](#)
- FREE SURFACE EFFECT COEFFICIENTS
 - in Cavity Expansion – in Layer, [576](#)
 - description of, [578](#)
- FRICTION COEFFICIENT
 - in Volume Repulsion Old, [594](#)
- FRICTION MODEL
 - in Contact Definition – in Interaction, [637](#)
 - description of, [640](#)
 - in Contact Definition – in Interaction Defaults, [634](#)
 - description of, [636](#)
- FRICTIONLESS MODEL
 - in Contact Definition
 - description of, [623](#)
- FULL TANGENT PRECONDITIONER
 - in CG, [140](#)
 - description of, [145](#), [153](#)
- FUNCTION, [67](#)
 - in Gravity, [558](#)
 - in Initial Condition, [489](#)

- in Periodic, [512](#)
- in Pore Pressure, [566](#)
 - description of, [567](#)
- in Prescribed Acceleration, [508](#)
 - description of, [510](#)
- in Prescribed Displacement, [499](#)
 - description of, [501](#)
- in Prescribed Force, [545](#)
 - description of, [547](#)
- in Prescribed Moment, [550](#)
 - description of, [552](#)
- in Prescribed Rotation, [516](#)
 - description of, [518](#)
- in Prescribed Rotational Velocity, [520](#)
 - description of, [522](#)
- in Prescribed Temperature, [561](#)
 - description of, [562](#)
- in Prescribed Velocity, [503](#)
 - description of, [505](#)
- in Pressure, [533](#)
 - description of, [535](#)
- in Traction, [541](#)
 - description of, [542](#)
- usage for thermal strains, [210](#)

G12

- in Elastic Orthotropic Fail material model, [307](#)
- in Elastic Orthotropic material model, [303](#)

G12_FUNCTION

- in Elastic Orthotropic Fail material model, [307](#)

G13

- in Elastic Orthotropic Fail material model, [307](#)
- in Elastic Orthotropic material model, [303](#)

G13_FUNCTION

- in Elastic Orthotropic Fail material model, [307](#)

G23

- in Elastic Orthotropic Fail material model, [307](#)
- in Elastic Orthotropic material model, [303](#)

G23_FUNCTION

- in Elastic Orthotropic Fail material model, [307](#)

GAMMA

- in Implicit Dynamics, [199](#)
- description of, [199](#)

GENERAL CONTACT

- in Contact Definition – in Interaction Defaults, [634](#)

GEOMETRIC POINT COORDINATES

- in Adaptive Refinement, [476](#)
- usage in, [476](#)

GLOBAL

- in Heartbeat Output – for global variables
 - description of, [736](#)
- in History Output – for global variables
 - description of, [729](#)

GLOBAL OPERATOR

- in User Variable, [831](#)
- GLOBAL OUTPUT OPTIONS
 - description of, [763](#)
- GLOBAL VARIABLES
 - in Procedural Transfer, [864](#)
 - in Results Output, [696](#)
 - description of, [706](#)
- GLUED MODEL
 - in Contact Definition
 - description of, [626](#)
- GRAVITATIONAL CONSTANT
 - in Fluid Pressure, [570](#)
 - in Fluid Pressure
 - usage in, [571](#)
 - in Gravity, [558](#)
- GRAVITY, [558](#)
- GROWTH FACTOR
 - in Adaptive Time Stepping
 - description of, [196](#)
- GXY
 - in Orthotropic Crush material model, [317](#)
- GYZ
 - in Orthotropic Crush material model, [317](#)
- GZX
 - in Orthotropic Crush material model, [317](#)
- HARDENING CONSTANT
 - in Barlat Plasticity material model, [258](#)
 - in Ductile Fracture material model, [231](#)
 - in Elastic-Plastic Power-Law Hardening material model, [228](#)
 - in Hill Plasticity material model, [252](#)
 - in Hosford Plasticity material model, [248](#)
 - in Plane Stress Rate Plasticity material model, [324](#)
- HARDENING EXPONENT
 - in Barlat Plasticity material model, [258](#)
 - in Ductile Fracture material model, [231](#)
 - in Elastic-Plastic Power-Law Hardening material model, [228](#)
 - in Hill Plasticity material model, [252](#)
 - in Hosford Plasticity material model, [248](#)
 - in Plane Stress Rate Plasticity material model, [324](#)
- HARDENING FUNCTION
 - in Barlat Plasticity material model, [258](#)
 - in Hill Plasticity material model, [252](#)
 - in Hosford Plasticity material model, [248](#)
 - in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
 - in Plane Stress Rate Plasticity material model, [324](#)
- HARDENING FUNCTIONS
 - in Thermoelastic Plastic material model, [244](#)
- HARDENING MODULUS
 - in Barlat Plasticity material model, [258](#)
 - in Elastic-Plastic material model, [224](#)
 - in Hill Plasticity material model, [252](#)
 - in Hosford Plasticity material model, [248](#)
 - in Plane Stress Rate Plasticity material model, [324](#)
 - in Thermoelastic Plastic material model, [244](#)
- HARDENING_EXPONENT_11
 - in Elastic Orthotropic Fail material model, [307](#)
- HARDENING_EXPONENT_12
 - in Elastic Orthotropic Fail material model, [307](#)
- HARDENING_EXPONENT_13
 - in Elastic Orthotropic Fail material model, [307](#)
- HARDENING_EXPONENT_22
 - in Elastic Orthotropic Fail material model, [307](#)
- HARDENING_EXPONENT_23
 - in Elastic Orthotropic Fail material model, [307](#)
- HARDENING_EXPONENT_33
 - in Elastic Orthotropic Fail material model, [307](#)
- HEARTBEAT OUTPUT, [734](#)
- HEIGHT
 - in Beam Section, [401](#)
- HISTORY OUTPUT, [728](#)
- HORIZON
 - in Peridynamics, [793](#)
 - in Peridynamics Section, [421](#)
- HOURLASS FORMULATION
 - in Solid Section, [378](#)
- HOURLASS INCREMENT
 - in Solid Section, [378](#)
- HOURLASS ROTATION
 - in Solid Section, [378](#)
- HOURLASS STIFFNESS
 - in Finite Element Model – in Parameters For Block, [363](#)
 - description of, [367](#)
 - in Peridynamics Section, [421](#)
 - with Peridynamics, [794](#)
- HOURLASS VISCOSITY
 - in Finite Element Model – in Parameters For Block, [363](#)
 - description of, [367](#)
- HYDRO EXPONENT
 - in Foam Plasticity material model, [289](#)
- HYDRO HARDENING
 - in Foam Plasticity material model, [289](#)
- HYDRO STRENGTH
 - in Foam Plasticity material model, [289](#)
- IDENTIFY FRAGMENTS, [784](#)
- IGNORE INITIAL OFFSET
 - in Spot Weld, [579](#)
- IMBALANCE TOLERANCE
 - in Zoltan Parameters, [470](#)

IMPLICIT DAMAGE SOLVER NUMBER OF ITERATIONS

in BCJ_MEM material model, 271

IMPLICIT DAMAGE SOLVER RESIDUAL TOLERANCE

in BCJ_MEM material model, 271

IMPLICIT DYNAMICS, 199

INACTIVE FOR PROCEDURE

in Finite Element Model – in Parameters For Block, 363

INACTIVE PERIODS, 93

description of, 93

in Adaptive Refinement, 476

usage in, 478

in Cavity Expansion, 576

in Centripetal Force, 560

in Element Death, 450

usage in, 463

in Fixed Displacement, 497

usage in, 498

in Fixed Rotation, 514

usage in, 515

in Fluid Pressure, 570

usage in, 572

in Gravity, 558

in Inertia Relief, 573

usage in, 574

in Line Weld, 585

in Mass Scaling, 128

usage in, 130

in Periodic, 512

in Pore Pressure, 566

usage in, 569

in Prescribed Acceleration, 508

usage in, 511

in Prescribed Displacement, 499

usage in, 503

in Prescribed Force, 545

usage in, 549

in Prescribed Moment, 550

usage in, 553

in Prescribed Rotation, 516

usage in, 520

in Prescribed Rotational Velocity, 520

usage in, 524

in Prescribed Temperature, 561

usage in, 564

in Prescribed Velocity, 503

usage in, 507

in Pressure, 533

in REFERENCE AXIS ROTATION

usage in, 527

in Silent Boundary, 578

in Spot Weld, 579

in Traction, 541

usage in, 544

in Viscous Damping, 590

usage in, 592

in Volume Repulsion Old

usage in, 595

in Volume Repulsion Old – in BLOCK SET, 594

INCLUDE

in Results Output, 696, 705

INCLUDE ALL BLOCKS

description of, 482

in Adaptive Refinement, 476

usage in, 477

in Centripetal Force, 560

in Element Death, 450

description of, 452

in Finite Element Model, 358

description of, 365

in Finite Element Model – in Parameters For Block, 363

in Fixed Displacement, 497

usage in, 498

in Fixed Rotation, 514

usage in, 515

in Gravity, 558

usage in, 558

in Inertia Relief, 573

usage in, 573

in Initial Condition, 489

usage in, 490

in Initial Velocity, 529

usage in, 530

in Mass Properties, 448

usage in, 448

in Mass Scaling, 128

usage in, 129

in Pore Pressure, 566

usage in, 567

in Prescribed Acceleration, 508

usage in, 509

in Prescribed Displacement, 499

usage in, 500

in Prescribed Force, 545

usage in, 547

in Prescribed Moment, 550

usage in, 551

in Prescribed Rotation, 516

usage in, 518

in Prescribed Rotational Velocity, 520

usage in, 522

in Prescribed Temperature, 561

usage in, 562

in Prescribed Velocity, 503

usage in, 505

- in Procedural Transfer, [864](#)
 - description of, [865](#)
- in Remesh Block Set, [475](#)
- in Time Step Initialization, [828](#)
 - usage in, [829](#)
- in User Output, [711](#)
 - usage in, [715](#)
- in Viscous Damping, [590](#)
 - usage in, [591](#)
- in Volume Repulsion Old
 - usage in, [595](#)
- in Volume Repulsion Old – in BLOCK SET, [594](#)
- INCLUDE NODES IN
 - in Rigid Body command block, [442](#)
- INCREASE OVER STEPS
 - in Lanczos Parameters, [116](#)
 - in Power Method Parameters, [123](#)
- INCREMENT INTERVAL
 - in Node Based Time Step Parameters, [126](#)
- INERTIA
 - in Rigid Body command block, [442](#)
- INERTIA RELIEF, [573](#)
- INFLUENCE FUNCTION
 - in Peridynamics Section, [421](#)
- Influence Function
 - with Peridynamics, [790](#)
- INITIAL ALPHA_XX
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL ALPHA_XY
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL ALPHA_XZ
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL ALPHA_YY
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL ALPHA_YZ
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL ALPHA_ZZ
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL CONDITION, [489](#)
- INITIAL DAMAGE
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL KAPPA
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
- INITIAL LOAD
 - in Truss Section, [411](#)
- INITIAL MESH MODIFICATION, [488](#)

- INITIAL REBALANCE
 - in Rebalance, [468](#)
 - description of, [469](#)
- INITIAL TEMPERATURE FOR UNCOUPLED ADIABATIC HEATING
 - in BCJ_MEM material model, [271](#)
- INITIAL TIME STEP
 - in Parameters For Presto Region, [103](#)
 - description of, [103](#)
- INITIAL VALUE
 - in User Variable, [831](#)
- INITIAL VELOCITY, [529](#)
- INITIAL ZETA
 - in BCJ_MEM material model, [271](#)
- INITIALIZE VARIABLE NAME
 - in Initial Condition, [489](#)
 - description of, [491](#)
- INPUT DATABASE NAME
 - in Restart Data, [745](#)
 - description of, [747](#)
- INTEGRATION RULE
 - in Shell Section, [388](#)
- INTERACTION
 - in Contact Definition
 - description of, [637](#)
- INTERACTION BEHAVIOR
 - in Contact Definition – in Interaction, [637](#)
 - description of, [640](#)
 - in Contact Definition – in Interaction Defaults, [634](#)
 - description of, [636](#)
- INTERACTION DEFAULTS
 - in Contact Definition
 - description of, [634](#)
- INTERFACE MATERIAL
 - in Contact Definition – in Interaction
 - description of, [640](#)
- INTERPOLATION TRANSFER
 - in Procedural Transfer, [864](#)
 - description of, [866](#)
- ISOTROPIC DYNAMIC RECOVERY CONSTANT
 - in BCJ_MEM material model, [271](#)
- ISOTROPIC DYNAMIC RECOVERY TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, [271](#)
- ISOTROPIC HARDENING CONSTANT
 - in BCJ_MEM material model, [271](#)
- ISOTROPIC HARDENING SHEAR COEFFICIENT
 - in BCJ_MEM material model, [271](#)
- ISOTROPIC HARDENING TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, [271](#)
- ISOTROPIC STATIC RECOVERY CONSTANT
 - in BCJ_MEM material model, [271](#)
- ISOTROPIC STATIC RECOVERY SINH CONSTANT

- in BCJ_MEM material model, [271](#)
- ISOTROPIC STATIC RECOVERY SINH TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, [271](#)
- ISOTROPIC STATIC RECOVERY TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, [271](#)
- ITERATION PLOT
 - in CG, [140](#)
 - description of, [152](#)
 - in Control Contact, [167](#)
 - in Control Damped Solve, [184](#)
 - in Control Failure, [181](#)
 - in Control Stiffness, [177](#)
- ITERATION PLOT OUTPUT BLOCKS
 - in CG, [140](#)
 - description of, [152](#)
 - in Control Contact, [167](#)
 - in Control Damped Solve, [184](#)
 - in Control Failure, [181](#)
 - in Control Stiffness, [177](#)
- ITERATION PRINT
 - in CG, [140](#)
 - description of, [152](#)
- ITERATION RESET
 - in CG, [140](#)
 - description of, [150](#)
- ITERATION UPDATE
 - in Full Tangent Preconditioner, [153](#)
 - description of, [157](#)
- ITERATION WINDOW
 - in Adaptive Time Stepping
 - description of, [195](#)
- K SCALING
 - in Incompressible Solid material model, [329](#)
- KEEP CUTS
 - in Zoltan Parameters, [470](#)
- L FUNCTION
 - in Orthotropic Rate material model, [321](#)
- L2_PROJECTION TRANSFER
 - in Procedural Transfer, [864](#)
 - description of, [866](#)
- LABELS
 - in Heartbeat Output, [734](#)
- LAGRANGE ADAPTIVE PENALTY
 - in Control Contact, [167](#)
 - description of, [171](#)
- LAGRANGE ADAPTIVE PENALTY GROWTH FACTOR
 - in Control Contact, [167](#)
 - description of, [171](#)

- LAGRANGE ADAPTIVE PENALTY REDUCTION FACTOR
 - in Control Contact, [167](#)
 - description of, [171](#)
- LAGRANGE ADAPTIVE PENALTY THRESHOLD
 - in Control Contact, [167](#)
 - description of, [171](#)
- LAGRANGE MAXIMUM PENALTY MULTIPLIER
 - in Control Contact, [167](#)
 - description of, [171](#)
- LAGRANGE MAXIMUM UPDATES
 - in Control Contact, [167](#)
 - description of, [172](#)
- LAGRANGE NODAL STIFFNESS MULTIPLIER
 - in Control Contact, [167](#)
 - description of, [172](#)
- LAGRANGE TOLERANCE
 - in Control Contact, [167](#)
 - description of, [172](#)
- LAMBDA
 - in Barlat Plasticity material model, [258](#)
 - in BCJ material model, [268](#)
 - in Ductile Fracture material model, [231](#)
 - in Elastic 3D Orthotropic material model, [297](#)
 - in Elastic Fracture material model, [222](#)
 - in Elastic material model, [217](#)
 - in Elastic Orthotropic Fail material model, [307](#)
 - in Elastic Orthotropic material model, [303](#)
 - in Elastic-Plastic material model, [224](#)
 - in Elastic-Plastic Power-Law Hardening material model, [228](#)
 - in Foam Plasticity material model, [289](#)
 - in Hill Plasticity material model, [252](#)
 - in Hosford Plasticity material model, [248](#)
 - in Hyperfoam material model, [295](#)
 - in Incompressible Solid material model, [329](#)
 - in Johnson–Cook material model, [265](#)
 - in Low Density Foam material model, [293](#)
 - in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
 - in Neo Hookean material model, [220](#)
 - in Orthotropic Crush material model, [317](#)
 - in Orthotropic Rate material model, [321](#)
 - in Plane Stress Rate Plasticity material model, [324](#)
 - in Power Law Creep material model, [279](#)
 - in Soil and Crushable Foam material model, [286](#)
 - in Stiff Elastic material model, [335](#)
 - in Swanson material model, [337](#)
 - in Thermoelastic material model, [219](#)
 - in Thermoelastic Plastic material model, [244](#)
 - in Viscoelastic Swanson material model, [340](#)
 - in Viscoplastic material model, [281](#)

- in Wire Mesh material model, [315](#)
- LANCZOS PARAMETERS, [116](#)
- LAYER
 - in Cavity Expansion, [576](#)
 - description of, [577](#)
- LAYER SURFACE
 - in Cavity Expansion – in Layer, [576](#)
 - description of, [577](#)
- LEGEND
 - in Heartbeat Output, [734](#)
- LEVEL
 - in Control Contact, [167](#)
 - in Control Failure, [181](#), [184](#)
 - in Control Stiffness, [177](#)
- LEVEL 1 PREDICTOR
 - in Solver, [135](#)
 - description of, [190](#)
- LINE CYLINDER RADIUS
 - in Volume Repulsion Old
 - usage in, [595](#)
 - in Volume Repulsion Old – in BLOCK SET, [594](#)
- LINE SEARCH
 - in CG, [140](#)
 - description of, [148](#)
- LINE WELD, [585](#)
- LINEAR BULK VISCOSITY
 - in Finite Element Model – in Parameters For Block, [363](#)
 - description of, [367](#)
- LINEAR SOLVER
 - in Full Tangent Preconditioner, [153](#)
 - description of, [155](#)
 - in Procedural Transfer, [864](#)
 - description of, [866](#)
- LOAD BALANCING METHOD
 - in Zoltan Parameters, [470](#)
 - as default, [470](#)
- LOAD RATIO THRESHOLD
 - in Rebalance, [468](#)
- LOADSTEP PREDICTOR
 - in Solver, [188](#)
- LOCAL SOLVER
 - in FETI Equation Solver, [162](#)
 - description of, [164](#)
- LOCALIZATION SECTION, [384](#)
- LOCK RCB DIRECTIONS
 - in Zoltan Parameters, [470](#)
- LOFTING FACTOR
 - in Membrane Section, [398](#)
 - in Shell Section, [388](#)
- LUDERS STRAIN
 - in Ductile Fracture material model, [231](#)
 - in Elastic-Plastic Power-Law Hardening material model, [228](#)
- LW FUNCTION
 - in Orthotropic Rate material model, [321](#)
- LX
 - in Orthotropic Rate material model, [321](#)
- LY
 - in Orthotropic Rate material model, [321](#)
- LZ
 - in Orthotropic Rate material model, [321](#)
- MAGNITUDE
 - in Initial Condition, [489](#)
 - description of, [491](#)
 - in Initial Velocity, [529](#)
 - description of, [530](#)
 - in Rigid Body command block, [442](#)
- MASS
 - in Rigid Body command block, [442](#)
- MASS DAMPING COEFFICIENT
 - in Viscous Damping, [590](#)
 - description of, [591](#)
- MASS PROPERTIES, [448](#)
- MASS SCALING, [128](#)
- MASTER
 - in Contact Definition – in Interaction, [637](#)
 - description of, [638](#)
 - in Periodic, [512](#)
- MASTER BLOCK
 - in MPC, [598](#)
- MASTER NODE SET
 - in MPC, [598](#)
- MASTER NODES
 - in MPC, [598](#)
- MASTER SLAVE MPC, [598](#)
- MASTER SURFACE
 - in MPC, [598](#)
- MATERIAL, [203](#)
 - in Finite Element Model – in Parameters For Block, [363](#)
 - description of, [365](#)
- MATERIAL CRITERION
 - in Element Death, [450](#)
 - description of, [456](#)
- MATERIAL MODEL FORMULATION
 - in Peridynamics Section, [421](#)
- MATRIX_STRENGTH_RATE_COEFFICIENT_11
 - in Elastic Orthotropic Fail material model, [307](#)
- MATRIX_STRENGTH_RATE_COEFFICIENT_12
 - in Elastic Orthotropic Fail material model, [307](#)
- MATRIX_STRENGTH_RATE_COEFFICIENT_13
 - in Elastic Orthotropic Fail material model, [307](#)
- MATRIX_STRENGTH_RATE_COEFFICIENT_22
 - in Elastic Orthotropic Fail material model, [307](#)
- MATRIX_STRENGTH_RATE_COEFFICIENT_23
 - in Elastic Orthotropic Fail material model, [307](#)

MATRIX_STRENGTH_RATE_COEFFICIENT_33
in Elastic Orthotropic Fail material model, 307

MAX NUMBER ELEMENTS
in Remesh, 472

MAX NUMBER ELEMENTS
in Remesh
usage in, 473

MAX POISSONS RATIO
in Incompressible Solid material model, 329
in Mooney-Rivlin material model, 332
in Swanson material model, 337
in Viscoelastic Swanson material model, 340

MAX REMESH REBALANCE METRIC
in Remesh, 472
usage in, 473

MAX REMESH STEP INTERVAL
in Remesh, 472
usage in, 473

MAX REMESH TIME INTERVAL
in Remesh, 472
usage in, 473

MAX STRESS
in Elastic Fracture material model, 222

MAX SUBINCREMENTS
in Power Law Creep material model, 279
in Viscoplastic material model, 281

MAXIMUM CHANGE IN DAMAGE ALLOWED PER
TIMESTEP
in BCJ_MEM material model, 271

MAXIMUM FAILURE CUTBACKS
in Adaptive Time Stepping
description of, 196

MAXIMUM ITERATIONS
in CG, 140
description of, 142
in Control Contact, 167
description of, 168
in Control Damped Solve, 184
description of, 185
in Control Failure, 181
description of, 182
in Control Stiffness, 177
description of, 178
in FETI Equation Solver, 162
description of, 163

MAXIMUM ITERATIONS FOR LOADSTEP
in Full Tangent Preconditioner, 153
description of, 157

MAXIMUM ITERATIONS FOR MODELPROBLEM
in Full Tangent Preconditioner, 153
description of, 157

MAXIMUM MULTIPLIER
in Adaptive Time Stepping
description of, 196

MAXIMUM ORTHOGONALIZATION
in FETI Equation Solver, 162
description of, 164

MAXIMUM RESETS FOR LOADSTEP
in Full Tangent Preconditioner, 153
description of, 157

MAXIMUM RESETS FOR MODELPROBLEM
in Full Tangent Preconditioner, 153
description of, 157

MAXIMUM SMOOTHING ITERATIONS
in Full Tangent Preconditioner, 153

MAXIMUM STEP LENGTH
in CG, 140
in CG
description of, 150

MAXIMUM_COMPRESSIVE_DAMAGE_11
in Elastic Orthotropic Fail material model, 307

MAXIMUM_COMPRESSIVE_DAMAGE_22
in Elastic Orthotropic Fail material model, 307

MAXIMUM_COMPRESSIVE_DAMAGE_33
in Elastic Orthotropic Fail material model, 307

MEMBRANE HOURGLASS STIFFNESS
in Finite Element Model – in Parameters For Block,
363
description of, 367

MEMBRANE HOURGLASS VISCOSITY
in Finite Element Model – in Parameters For Block,
363
description of, 367

MEMBRANE SECTION, 398

MESH QUALITY, 90

METHOD
in Adaptive Time Stepping
description of, 194

MINIMUM CONVERGENCE RATE
in Full Tangent Preconditioner, 153
description of, 160

MINIMUM ITERATIONS
in CG, 140
description of, 142
in Control Contact, 167
description of, 168
in Control Stiffness, 177
description of, 178

MINIMUM MULTIPLIER
in Adaptive Time Stepping
description of, 197

MINIMUM RESIDUAL IMPROVEMENT
in CG, 140
description of, 142

MINIMUM SMOOTHING ITERATIONS
in Full Tangent Preconditioner, 153

MINIMUM STEP LENGTH
in CG, 140

- in CG
 - description of, [150](#)
- MISORIENTATION VARIABLE HARDENING CONSTANT
 - in BCJ_MEM material model, [271](#)
- MISORIENTATION VARIABLE HARDENING EXPONENT
 - in BCJ_MEM material model, [271](#)
- MODEL
 - in Finite Element Model – in Parameters For Block, [363](#)
 - description of, [365](#)
- MODULUS FUNCTION
 - in Orthotropic Rate material model, [321](#)
- MODULUS LLLL
 - in Orthotropic Rate material model, [321](#)
- MODULUS LLWW
 - in Orthotropic Rate material model, [321](#)
- MODULUS LWLW
 - in Orthotropic Rate material model, [321](#)
- MODULUS TLTL
 - in Orthotropic Rate material model, [321](#)
- MODULUS TTLL
 - in Orthotropic Rate material model, [321](#)
- MODULUS TTTT
 - in Orthotropic Rate material model, [321](#)
- MODULUS TTWW
 - in Orthotropic Rate material model, [321](#)
- MODULUS WTWL
 - in Orthotropic Rate material model, [321](#)
- MODULUS WWWW
 - in Orthotropic Rate material model, [321](#)
- MOMENTUM BALANCE ITERATIONS
 - in Contact Definition – in Enforcement Options, [660](#)
 - description of, [661](#)
- MONAGHAN EPSILON
 - in Particle Section, [417](#)
- MONAGHAN N
 - in Particle Section, [417](#)
- MONITOR
 - in Heartbeat Output, [734](#)
 - description of, [743](#)
- MOUNT TYPE
 - in User Mount Section, [424](#)
- MOVE BLOCK
 - in Initial Mesh Modification, [488](#)
- N
 - in Hyperfoam material model, [295](#)
- NAIR
 - in Low Density Foam material model, [293](#)
- NEAREST ELEMENT COPY
 - in Procedural Transfer, [864](#)
 - description of, [866](#)
- NEW MATERIAL
 - in Particle Conversion, [462](#)
- NEW MESH MAX EDGE LENGTH RATIO
 - in Remesh, [472](#)
 - usage in, [473](#)
- NEW MESH MIN EDGE LENGTH RATIO
 - in Remesh, [472](#)
 - usage in, [473](#)
- NEW MESH MIN SHAPE
 - in Remesh, [472](#)
 - usage in, [473](#)
- NODAL
 - in Heartbeat Output, [734](#)
 - in Heartbeat Output – for mesh entities variables description of, [737](#)
 - in Heartbeat Output – for nearest point variables description of, [738](#)
 - in History Output, [728](#)
 - about, [729](#), [736](#)
 - in History Output – for mesh entities variables description of, [730](#)
 - in History Output – for nearest point variables description of, [731](#)
- NODAL DIAGONAL SCALE
 - in CG, [140](#)
 - description of, [145](#)
- NODAL DIAGONAL SHIFT
 - in CG, [140](#)
 - description of, [145](#)
- NODAL PRECONDITIONER METHOD
 - in Full Tangent Preconditioner, [153](#)
 - description of, [155](#)
- NODAL PROBE FACTOR
 - in CG, [140](#)
 - description of, [145](#)
- NODAL VARIABLES
 - in Results Output, [696](#)
 - description of, [698](#)
- NODE
 - in Heartbeat Output, [734](#)
 - in Heartbeat Output – for mesh entities variables description of, [737](#)
 - in Heartbeat Output – for nearest point variables description of, [738](#)
 - in History Output, [728](#)
 - about, [729](#), [736](#)
 - in History Output – for mesh entities variables description of, [730](#)
 - in History Output – for nearest point variables description of, [731](#)
 - in Inertia Relief, [573](#)
 - usage in, [573](#)
- NODE BASED ALPHA FACTOR
 - in Solid Section, [378](#)

NODE BASED BETA FACTOR

in Solid Section, [378](#)

NODE BASED STABILIZATION METHOD

in Solid Section, [378](#)

NODE BASED TIME STEP PARAMETERS, [126](#)

NODE SET

description of, [482](#)

in Adaptive Refinement, [476](#)

usage in, [477](#)

in Centripetal Force, [560](#)

in Contact Definition – in Contact Surface (block)

usage in, [612](#)

in Fixed Displacement, [497](#)

usage in, [498](#)

in Fixed Rotation, [514](#)

usage in, [515](#)

in Gravity, [558](#)

usage in, [558](#)

in Heartbeat Output, [734](#)

in Heartbeat Output – for mesh entities variables

description of, [737](#)

in Heartbeat Output – for nearest point variables

description of, [738](#)

in History Output, [728](#)

about, [729](#), [736](#)

in History Output – for mesh entities variables

description of, [730](#)

in History Output – for nearest point variables

description of, [731](#)

in Inertia Relief, [573](#)

usage in, [573](#)

in Initial Condition, [489](#)

usage in, [490](#)

in Initial Velocity, [529](#)

usage in, [530](#)

in Mass Scaling, [128](#)

usage in, [129](#)

in Prescribed Acceleration, [508](#)

usage in, [509](#)

in Prescribed Displacement, [499](#)

usage in, [500](#)

in Prescribed Force, [545](#)

usage in, [547](#)

in Prescribed Moment, [550](#)

usage in, [551](#)

in Prescribed Rotation, [516](#)

usage in, [518](#)

in Prescribed Rotational Velocity, [520](#)

usage in, [522](#)

in Prescribed Velocity, [503](#)

usage in, [505](#)

in Spot Weld, [579](#)

in Time Step Initialization, [828](#)

usage in, [829](#)

in User Output, [711](#)

usage in, [715](#)

NODE SET SUBROUTINE

as user subroutine command line, [826](#)

description of, [826](#)

in Initial Condition, [489](#)

description of, [494](#)

in Initial Velocity, [529](#)

description of, [532](#)

in Pore Pressure

description of, [567](#)

in Prescribed Acceleration, [508](#)

description of, [511](#)

in Prescribed Displacement, [499](#)

description of, [502](#)

in Prescribed Force, [545](#)

description of, [548](#)

in Prescribed Moment, [550](#)

description of, [552](#)

in Prescribed Rotation, [516](#)

description of, [519](#)

in Prescribed Rotational Velocity, [520](#)

description of, [523](#)

in Prescribed Temperature, [561](#)

description of, [563](#)

in Prescribed Velocity, [503](#)

description of, [507](#)

in Pressure, [533](#)

description of, [536](#)

in Time Step Initialization, [828](#)

description of, [830](#)

in Traction, [541](#)

description of, [543](#)

in User Output, [711](#)

description of, [721](#)

NODE SET VARIABLES

in Results Output, [696](#)

description of, [699](#)

NODE SETS TO DEFINE BODY AXIS

in Cavity Expansion, [576](#)

NODE VARIABLES

in Procedural Transfer, [864](#)

description of, [865](#)

in Results Output, [696](#)

description of, [698](#)

NODESET

in Heartbeat Output, [734](#)

in Heartbeat Output – for mesh entities variables

description of, [737](#)

in Heartbeat Output – for nearest point variables

description of, [738](#)

in History Output, [728](#)

about, [729](#), [736](#)

in History Output – for mesh entities variables

- description of, 730
- in History Output – for nearest point variables
 - description of, 731
- NODESET VARIABLES
 - in Results Output, 696
 - description of, 699
- NORMAL DISPLACEMENT FUNCTION
 - in Spot Weld, 579
- NORMAL DISPLACEMENT SCALE FACTOR
 - in Spot Weld, 579
- NORMAL TOLERANCE
 - in Contact Definition – in Interaction, 637
 - description of, 641
 - in Contact Definition – in Search Options, 659
 - description of, 659
- NU12
 - in Elastic Orthotropic Fail material model, 307
 - in Elastic Orthotropic material model, 303
- NU12_FUNCTION
 - in Elastic Orthotropic Fail material model, 307
- NU13
 - in Elastic Orthotropic Fail material model, 307
 - in Elastic Orthotropic material model, 303
- NU13_FUNCTION
 - in Elastic Orthotropic Fail material model, 307
- NU23
 - in Elastic Orthotropic Fail material model, 307
 - in Elastic Orthotropic material model, 303
- NU23_FUNCTION
 - in Elastic Orthotropic Fail material model, 307
- NUM LOCAL SUBDOMAINS
 - in FETI Equation Solver, 162
 - description of, 164
- NUMBER EIGENVALUES
 - in Lanczos Parameters, 116
- NUMBER ITERATIONS
 - in Power Method Parameters, 123
- NUMBER OF INTEGRATION POINTS
 - in Cohesive Section, 384
 - in Localization Section, 384
 - in Shell Section, 388
- NUMBER OF TIME STEPS
 - in Parameters For Adagio Region, 192, 193
- NUMERICAL SHIFT FUNCTION
 - in Viscoelastic Swanson material model, 340
- OBJECT TYPE
 - in Pressure, 533
 - description of, 537
- OMIT BLOCK
 - in Finite Element Model, 358
 - description of, 362
- OPTIONAL
 - in Restart Data, 745
- description of, 747
- ORDINATE
 - in function, 67
- ORIENTATION, 80
 - in Shell Section, 388
- ORIGIN
 - in Coordinate System, 84
- ORIGIN NODE
 - in Coordinate System, 84
- ORTHOGONALITY MEASURE FOR RESET
 - in CG, 140
 - description of, 150
- OUTPUT DATABASE NAME
 - in Restart Data, 745
 - description of, 747
- OUTPUT MESH
 - EXPOSED SURFACE, 706
 - in Results Output, 696
 - description of, 706
 - REFINED, 706
- OUTPUT ON SIGNAL
 - in Heartbeat Output, 734
 - description of, 740
 - in History Output, 728
 - description of, 733
 - in Restart Data, 745
 - description of, 755
 - in Results Output, 696
 - description of, 710
- OUTPUT SCHEDULER, 757
 - example of, 758
 - use of, 757
- OVER ALLOCATE MEMORY
 - in Zoltan Parameters, 470
 - as default, 470
- OVERLAP ITERATIONS
 - in Contact Definition – in Remove Initial Overlap
 - description of, 651
- OVERLAP NORMAL TOLERANCE
 - in Contact Definition – in Interaction, 637
 - description of, 641
 - in Contact Definition – in Remove Initial Overlap
 - description of, 651
- OVERLAP TANGENTIAL TOLERANCE
 - in Contact Definition – in Interaction, 637
 - description of, 641
 - in Contact Definition – in Remove Initial Overlap
 - description of, 651
- OVERLAY COUNT
 - in Restart Data, 745
 - description of, 754
- OVERWRITE
 - in History Output, 728
 - in Restart Data, 745

- in Results Output, [696](#)
- P0
 - in Low Density Foam material model, [293](#)
- P1
 - in Swanson material model, [337](#)
 - in Viscoelastic Swanson material model, [340](#)
- PARAM-STRING debugMask VALUE
 - in FETI Equation Solver, [162](#)
- PARAM-STRING precision VALUE
 - in FETI Equation Solver, [162](#)
 - description of, [164](#)
- PARAMETERS
 - in User Mount Section, [424](#)
- PARAMETERS FOR ADAGIO REGION
 - in Time Stepping Block, [191](#)
- PARAMETERS FOR BLOCK
 - in Finite Element Model, [358](#)
 - about, [364](#)
 - listing of, [363](#)
- PARAMETERS FOR MODEL
 - in material command blocks
 - description of, [204](#)
- PARAMETERS FOR MODEL BARLAT_PLASTICITY
 - in Barlat Plasticity material model, [258](#)
- PARAMETERS FOR MODEL BCJ
 - in BCJ material model, [268](#)
- PARAMETERS FOR MODEL BCJ_MEM
 - in BCJ_MEM material model, [271](#)
- PARAMETERS FOR MODEL DUCTILE FRACTURE
 - in Ductile Fracture material model, [231](#)
- PARAMETERS FOR MODEL ELASTIC
 - in Elastic material model, [217](#)
 - in Neo Hookean material model, [220](#)
 - in Thermoelastic material model, [219](#)
- PARAMETERS FOR MODEL
 - ELASTIC_3D_ORTHOTROPIC
 - in Elastic 3D Orthotropic material model, [297](#)
- PARAMETERS FOR MODEL ELASTIC_FRACTURE
 - in Elastic Fracture material model, [222](#)
- PARAMETERS FOR MODEL
 - ELASTIC_ORTHOTROPIC
 - in Elastic Orthotropic Fail material model, [307](#)
 - in Elastic Orthotropic material model, [303](#)
- PARAMETERS FOR MODEL ELASTIC_PLASTIC
 - in Elastic-Plastic material model, [224](#)
- PARAMETERS FOR MODEL EP_POWER_HARD
 - in Elastic-Plastic Power-Law Hardening material model, [228](#)
- PARAMETERS FOR MODEL FOAM_PLASTICITY
 - in Foam Plasticity material model, [289](#)
- PARAMETERS FOR MODEL HILL_PLASTICITY
 - in Hill Plasticity material model, [252](#)
- PARAMETERS FOR MODEL
 - HOSFORD_PLASTICITY
 - in Hosford Plasticity material model, [248](#)
- PARAMETERS FOR MODEL HYPERFOAM
 - in Hyperfoam material model, [295](#)
- PARAMETERS FOR MODEL
 - INCOMPRESSIBLE_SOLID
 - in Incompressible Solid material model, [329](#)
- PARAMETERS FOR MODEL JOHNSON_COOK
 - in Johnson-Cook material model, [265](#)
- PARAMETERS FOR MODEL
 - LOW_DENSITY_FOAM
 - in Low Density Foam material model, [293](#)
- PARAMETERS FOR MODEL ML_EP_FAIL
 - in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material mode, [238](#)
- PARAMETERS FOR MODEL MOONEY_RIVLIN
 - in Mooney-Rivlin material model, [332](#)
- PARAMETERS FOR MODEL
 - ORTHOTROPIC_CRUSH
 - in Orthotropic Crush material model, [317](#)
- PARAMETERS FOR MODEL ORTHOTROPIC_RATE
 - in Orthotropic Rate material model, [321](#)
- PARAMETERS FOR MODEL
 - PLANE_STRESS_RATE_PLASTICITY
 - in Plane Stress Rate Plasticity material model, [324](#)
- PARAMETERS FOR MODEL POWER_LAW_CREEP
 - in Power Law Creep material model, [279](#)
 - in Viscoplastic material model, [281](#)
- PARAMETERS FOR MODEL SOIL_FOAM
 - in Soil and Crushable Foam material model, [286](#)
- PARAMETERS FOR MODEL STIFF_ELASTIC
 - in Stiff Elastic material model, [335](#)
- PARAMETERS FOR MODEL SWANSON
 - in Swanson material model, [337](#)
- PARAMETERS FOR MODEL THERMO ELASTIC PLASTIC
 - in Thermoelastic Plastic material model, [244](#)
- PARAMETERS FOR MODEL
 - VISCOELASTIC_SWANSON
 - in Viscoelastic Swanson material model, [340](#)
- PARAMETERS FOR MODEL WIRE_MESH
 - in Wire Mesh material model, [315](#)
- PARAMETERS FOR PRESTO REGION
 - in Time Stepping Block, [102](#)
- PARAMETERS FOR PRESTO REGION
 - in Time Stepping Block
 - contents of, [103](#)
- PARAMETERS VARIABLE
 - in User Mount Section, [424](#)
- PARTICLE CONVERSION
 - in Element Death, [450](#)

- description of, 462
- PARTICLE EMBEDDING, 466
- PARTICLE SECTION, 417
 - in Particle Conversion, 462
- PENALTY FACTOR
 - in Full Tangent Preconditioner, 153
 - description of, 155
- Peridynamics
 - in Special Modeling Techniques, 787
- PERIDYNAMICS SECTION, 421
- PERIOD
 - in Truss Section, 411
- PERIODIC, 512
- PERIODIC REBALANCE
 - in Rebalance, 468
 - description of, 469
- PHI
 - in Foam Plasticity material model, 289
 - in Low Density Foam material model, 293
- PLASTIC DISSIPATION FACTOR
 - in BCJ_MEM material model, 271
- POINT
 - in Coordinate System, 84
- POINT A
 - in Orientation, 80
- POINT B
 - in Orientation, 80
- POINT INERTIA
 - in Rigid Body command block, 442
- POINT MASS
 - in Rigid Body command block, 442
- POINT MASS SECTION, 414
- POINT NODE
 - in Coordinate System, 84
- POINT ON AXIS
 - in Periodic, 512
- POISSON
 - in Hyperfoam material model, 295
- POISSONS RATIO
 - in Barlat Plasticity material model, 258
 - in BCJ material model, 268
 - in BCJ_MEM model, 271
 - in Ductile Fracture material model, 231
 - in Elastic 3D Orthotropic material model, 297
 - in Elastic Fracture material model, 222
 - in Elastic material model, 217
 - in Elastic Orthotropic Fail material model, 307
 - in Elastic Orthotropic material model, 303
 - in Elastic-Plastic material model, 224
 - in Elastic-Plastic Power-Law Hardening material model, 228
 - in Foam Plasticity material model, 289
 - in Hill Plasticity material model, 252
 - in Hosford Plasticity material model, 248
 - in Hyperfoam material model, 295
 - in Incompressible Solid material model, 329
 - in Johnson–Cook material model, 265
 - in Low Density Foam material model, 293
 - in Multilinear Elastic-Plastic Hardening Model material model, 234
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, 238
 - in Neo Hookean material model, 220
 - in Orthotropic Crush material model, 317
 - in Orthotropic Rate material model, 321
 - in Plane Stress Rate Plasticity material model, 324
 - in Power Law Creep material model, 279
 - in Soil and Crushable Foam material model, 286
 - in Stiff Elastic material model, 335
 - in Swanson material model, 337
 - in Thermoelastic material model, 219
 - in Thermoelastic Plastic material model, 244
 - in Viscoelastic Swanson material model, 340
 - in Viscoplastic material model, 281
 - in Wire Mesh material model, 315
- POISSONS RATIO AB
 - in Elastic 3D Orthotropic material model, 297
- POISSONS RATIO BC
 - in Elastic 3D Orthotropic material model, 297
- POISSONS RATIO CA
 - in Elastic 3D Orthotropic material model, 297
- POISSONS RATIO FUNCTION
 - in BCJ material model, 268
 - in BCJ_MEM material model, 271
 - in Multilinear Elastic-Plastic Hardening Model material model, 234
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, 238
 - in Thermoelastic material model, 219
 - in Thermoelastic Plastic material model, 244
- PORE PRESSURE, 566
- POWER METHOD PARAMETERS, 123
- PRECISION
 - in Heartbeat Output, 734
- PRECONDITIONER
 - in CG, 140
 - description of, 145
- PRECONDITIONER ITERATION UPDATE
 - in CG, 140
 - description of, 145
- PRECONDITIONING METHOD
 - in FETI Equation Solver, 162
 - description of, 164
- PREDICTOR TYPE
 - in Loadstep Predictor
 - description of, 188
- PRESCRIBED ACCELERATION, 508
- PRESCRIBED DISPLACEMENT, 499

PRESCRIBED FORCE, [545](#)
 PRESCRIBED MOMENT, [550](#)
 PRESCRIBED QUANTITY
 in Periodic, [512](#)
 PRESCRIBED ROTATION, [516](#)
 PRESCRIBED ROTATIONAL Velocity, [520](#)
 PRESCRIBED TEMPERATURE, [561](#)
 usage for thermal strains, [211](#)
 PRESCRIBED VELOCITY, [503](#)
 PRESSURE, [533](#)
 PRESSURE COEFFICIENTS
 in Cavity Expansion – in Layer, [576](#)
 description of, [577](#)
 PRESSURE CUTOFF
 in Soil and Crushable Foam material model, [286](#)
 PRESSURE FUNCTION
 in Soil and Crushable Foam material model, [286](#)
 PRESSURE INTEGRATION OPTION
 in BCJ_MEM material model, [271](#)
 PRESTO PROCEDURE, [87](#)
 description of, [88](#)
 PRESTO REGION
 in Presto Procedure, [87](#)
 description of, [89](#)
 PROBE FACTOR
 in Full Tangent Preconditioner, [153](#)
 description of, [155](#)
 PROBLEM DIMENSION
 in Particle Section, [417](#)
 PROCEDURAL TRANSFER, [864](#)
 PRONY SHEAR 1
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 10
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 2
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 3
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 4
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 5
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 6
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 7
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 8
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR 9
 in Viscoelastic Swanson material model, [340](#)
 PRONY SHEAR INFINITY
 in Viscoelastic Swanson material model, [340](#)
 PUSHBACK FACTOR
 in Contact Definition – in Interaction, [637](#)
 description of, [641](#)
 Q1
 in Swanson material model, [337](#)
 in Viscoelastic Swanson material model, [340](#)
 Q1P0 STABILIZATION THRESHOLD
 in Solid Section, [378](#)
 Q1P0 TIMESTEP LENGTH SCALE
 in Solid Section, [378](#)
 Q1P0 TIMESTEP SCALE FACTOR
 in Solid Section, [378](#)
 Q1P0 TIMESTEP WAVE SPEED
 in Solid Section, [378](#)
 QUADRATIC BULK VISCOSITY
 in Finite Element Model – in Parameters For Block, [363](#)
 description of, [367](#)
 R DISPLACEMENT FUNCTION
 in Line Weld, [585](#)
 R DISPLACEMENT SCALE FACTOR
 in Line Weld, [585](#)
 R ROTATION FUNCTION
 in Line Weld, [585](#)
 R ROTATION SCALE FACTOR
 in Line Weld, [585](#)
 R1
 in Swanson material model, [337](#)
 in Viscoelastic Swanson material model, [340](#)
 R11
 in Hill Plasticity material model, [252](#)
 R12
 in Hill Plasticity material model, [252](#)
 R22
 in Hill Plasticity material model, [252](#)
 R23
 in Hill Plasticity material model, [252](#)
 R31
 in Hill Plasticity material model, [252](#)
 R33
 in Hill Plasticity material model, [252](#)
 RADIAL AXIS
 in Prescribed Displacement, [499](#)
 description of, [501](#)
 in Prescribed Velocity, [503](#)
 description of, [505](#)
 RADIUS
 in Adaptive Refinement, [476](#)
 usage in, [476](#)
 RADIUS MESH VARIABLE
 in Particle Section, [417](#)
 RADIUS TIME STEP
 in Particle Section, [417](#)
 RATE CONSTANT

- in Plane Stress Rate Plasticity material model, 324
- RATE FUNCTION
 - in Orthotropic Rate material model, 321
 - in Plane Stress Rate Plasticity material model, 324
- RATE INDEPENDENT YIELD CONSTANT
 - in BCJ_MEM material model, 271
- RATE INDEPENDENT YIELD SHEAR COEFFICIENT
 - in BCJ_MEM material model, 271
- RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE
 - in BCJ_MEM material model, 271
- RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE2
 - in BCJ_MEM material model, 271
- RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE3
 - in BCJ_MEM material model, 271
- RATE INDEPENDENT YIELD TEMPERATURE DEPENDENCE4
 - in BCJ_MEM material model, 271
- RBE3, 601
- READ VARIABLE
 - in Initial Condition, 489
 - in Pore Pressure, 566
 - description of, 568
 - in Prescribed Acceleration, 508
 - description of, 511
 - in Prescribed Displacement, 499
 - description of, 503
 - in Prescribed Rotation, 516
 - description of, 519
 - in Prescribed Rotational Velocity, 520
 - description of, 524
 - in Prescribed Temperature, 561
 - description of, 563
 - in Prescribed Velocity, 503
 - description of, 507
 - in Pressure, 533
 - description of, 537
 - in Time Step Initialization
 - description of, 830
- REBALANCE, 468
- REBALANCE STEP INTERVAL
 - in Rebalance, 468
- RECEIVE BLOCKS
 - in Procedural Transfer, 864
 - description of, 866
- RECEIVE COORDINATES
 - in Procedural Transfer, 864
 - description of, 866
- RECEIVE FROM TRANSFER
 - in Pore Pressure, 566
 - in Prescribed Temperature, 561
- RECTILINEAR RCB BLOCKS
 - in Zoltan Parameters, 470
- REFERENCE
 - in CG, 140
 - description of, 142
 - in Control Stiffness, 177
 - description of, 178
- REFERENCE AXIS, 524
 - in Beam Section, 401
 - in Periodic, 512
- REFERENCE AXIS X FUNCTION
 - in Reference Axis, 524
 - description of, 526
- REFERENCE AXIS Y FUNCTION
 - in Reference Axis, 524
 - description of, 526
- REFERENCE AXIS Z FUNCTION
 - in Reference Axis, 524
 - description of, 526
- REFERENCE BLOCK
 - in RBE3, 601
- REFERENCE LOCATION
 - in Rigid Body command block, 442
- REFERENCE NODE SET
 - in RBE3, 601
- REFERENCE NODES
 - in RBE3, 601
- REFERENCE POINT
 - in Fluid Pressure, 570
 - in Fluid Pressure
 - usage in, 571
- REFERENCE STRAIN
 - in Incompressible Solid material model, 329
 - in Mooney-Rivlin material model, 332
 - in Stiff Elastic material model, 335
 - in Swanson material model, 337
 - in Viscoelastic Swanson material model, 340
- REFERENCE SURFACE
 - in RBE3, 601
- REFERENCE_STRAIN_RATE
 - in Elastic Orthotropic Fail material model, 307
- REMESH, 472
- REMESH AT MAX EDGE LENGTH RATIO
 - in Remesh, 472
 - usage in, 473
- REMESH AT MIN EDGE LENGTH RATIO
 - in Remesh, 472
 - usage in, 473
- REMESH AT SHAPE
 - in Remesh, 472
 - usage in, 473
- REMESH BLOCK SET, 475
 - in Remesh, 472
- REMOVE BLOCK

- description of, 484
- in Adaptive Refinement, 476
 - usage in, 477
- in Centripetal Force, 560
- in Contact Definition –in Contact Surface (block)
 - usage in, 612
- in Element Death, 450
 - description of, 452
- in Finite Element Model, 358
 - description of, 366
- in Finite Element Model – in Parameters For Block, 363
- in Fixed Displacement, 497
 - usage in, 498
- in Fixed Rotation, 514
 - usage in, 515
- in Gravity, 558
 - usage in, 558
- in Inertia Relief, 573
 - usage in, 573
- in Initial Condition, 489
 - usage in, 490
- in Initial Velocity, 529
 - usage in, 530
- in Line Weld, 585
- in Mass Properties, 448
 - usage in, 448
- in Mass Scaling, 128
 - usage in, 129
- in Pore Pressure, 566
 - usage in, 567
- in Prescribed Acceleration, 508
 - usage in, 509
- in Prescribed Displacement, 499
 - usage in, 500
- in Prescribed Force, 545
 - usage in, 547
- in Prescribed Moment, 550
 - usage in, 551
- in Prescribed Rotation, 516
 - usage in, 518
- in Prescribed Rotational Velocity, 520
 - usage in, 522
- in Prescribed Temperature, 561
 - usage in, 562
- in Prescribed Velocity, 503
 - usage in, 505
- in Procedural Transfer, 864
 - description of, 865
- in Remesh Block Set, 475
- in Time Step Initialization, 828
 - usage in, 829
- in User Output, 711
 - usage in, 715
- in Viscous Damping, 590
 - usage in, 591
- in Volume Repulsion Old
 - usage in, 595
- in Volume Repulsion Old – in BLOCK SET, 594

REMOVE GAPS AND OVERLAPS

- in MPC, 598
- in TIED MPC, 595

REMOVE INITIAL OVERLAP

- in Contact Definition
 - description of, 651

REMOVE NODE SET

- description of, 484
- in Adaptive Refinement, 476
 - usage in, 477
- in Centripetal Force, 560
- in Contact Definition – in Contact Surface (block)
 - usage in, 612
- in Fixed Displacement, 497
 - usage in, 498
- in Fixed Rotation, 514
 - usage in, 515
- in Gravity, 558
 - usage in, 558
- in Inertia Relief, 573
 - usage in, 573
- in Initial Condition, 489
 - usage in, 490
- in Initial Velocity, 529
 - usage in, 530
- in Mass Scaling, 128
 - usage in, 129
- in Prescribed Acceleration, 508
 - usage in, 509
- in Prescribed Displacement, 499
 - usage in, 500
- in Prescribed Force, 545
 - usage in, 547
- in Prescribed Moment, 550
 - usage in, 551
- in Prescribed Rotation, 516
 - usage in, 518
- in Prescribed Rotational Velocity, 520
 - usage in, 522
- in Prescribed Velocity, 503
 - usage in, 505
- in Spot Weld, 579
- in Time Step Initialization, 828
 - usage in, 829
- in User Output, 711
 - usage in, 715

REMOVE SURFACE

- description of, 484
- in Adaptive Refinement, 476

- usage in, 477
- in Cavity Expansion, 576
- in Centripetal Force, 560
- in Contact Definition – in Contact Surface (block)
 - usage in, 612
- in Fixed Displacement, 497
 - usage in, 498
- in Fixed Rotation, 514
 - usage in, 515
- in Gravity, 558
 - usage in, 558
- in Inertia Relief, 573
 - usage in, 573
- in Initial Condition, 489
 - usage in, 490
- in Initial Velocity, 529
 - usage in, 530
- in Line Weld , 585
- in Mass Scaling, 128
 - usage in, 129
- in Prescribed Acceleration, 508
 - usage in, 509
- in Prescribed Displacement, 499
 - usage in, 500
- in Prescribed Force, 545
 - usage in, 547
- in Prescribed Moment, 550
 - usage in, 551
- in Prescribed Rotation, 516
 - usage in, 518
- in Prescribed Rotational Velocity, 520
 - usage in, 522
- in Prescribed Velocity, 503
 - usage in, 505
- in Pressure, 533
- in Silent Boundary, 578
- in Spot Weld, 579
- in Time Step Initialization, 828
 - usage in, 829
- in Traction, 541
- in User Output, 711
 - usage in, 715
- in Volume Repulsion Old
 - usage in, 595
- in Volume Repulsion Old – in BLOCK SET, 594

RENUMBER PARTITIONS

- in Zoltan Parameters, 470

RESET AT NEW PERIOD

- in Adaptive Time Stepping
 - description of, 197

RESET LIMITS

- in CG, 140
 - description of, 150

RESIDUAL NORM TOLERANCE

- in FETI Equation Solver, 162
 - description of, 163

RESIDUAL NORM TYPE

- in CG, 140
 - description of, 142

RESIDUAL ROUNDOFF TOLERANCE

- in CG
 - description of, 142
- in CG, 140

RESOLVE MULTIPLE MPCs, 598

RESTART, 66

- about, 66

RESTART DATA, 745

- about, 66
- about auto read and write, 748
- about overwriting, 752
- about recovering, 753
- about user-controlled read and write, 750
- Implicit/Explicit Restart, 756

RESTART TIME, 66

- about, 66

RESULTS OUTPUT, 696

REUSE CUTS

- in Zoltan Parameters, 470
 - as default, 470

RHO

- in BCJ material model, 268

RIGID BODIES FROM ATTRIBUTES

- in Beam Section, 401
- in Membrane Section, 398
- in Point Mass Section, 414
- in Shell Section, 388
- in Solid Section, 378
- in Truss Section, 411

RIGID BODY

- in Fixed Displacement, 497
 - usage in, 498
- in Initial Velocity, 529
- in Prescribed Acceleration, 508
- in Prescribed Displacement, 499
- in Prescribed Rotation
 - usage in, 518
- in Reference Axis Rotation
 - usage in, 526
- as command block, 442
- in Beam Section, 401
- in Fixed Rotation, 514
 - usage in, 515
- in Initial Velocity
 - usage in, 530
- in Membrane Section, 398
- in Point Mass Section, 414
- in Prescribed Acceleration
 - usage in, 509

- in Prescribed Displacement
 - usage in, [500](#)
- in Prescribed Force, [545](#)
 - usage in, [547](#)
- in Prescribed Moment, [550](#)
 - usage in, [551](#)
- in Prescribed Rotation, [516](#)
- in Prescribed Rotational Velocity, [520](#)
 - usage in, [522](#)
- in Prescribed Velocity, [503](#)
 - usage in, [505](#)
- in Reference Axis, [524](#)
- in Shell Section, [388](#)
- in Solid Section, [378](#)
- in Truss Section, [411](#)

ROTATE BLOCK

- in Initial Mesh Modification, [488](#)

ROTATION

- in Reference Axis, [524](#)
- in Reference Axis Rotation
 - description of, [527](#)

ROTATION ABOUT

- in Orientation, [80](#)

ROTATIONAL SCALE FACTOR

- in Centripetal Force, [560](#)

ROTATIONAL VELOCITY

- in Reference Axis, [524](#)
- in Reference Axis Rotation
 - description of, [527](#)

ROTATIONAL VELOCITY FUNCTION

- in Centripetal Force, [560](#)

ROTATION_AXIS_1

- in Elastic Orthotropic Fail material model, [307](#)
- in Elastic Orthotropic material model, [303](#)

ROTATION_AXIS_2

- in Elastic Orthotropic Fail material model, [307](#)
- in Elastic Orthotropic material model, [303](#)

ROTATION_AXIS_3

- in Elastic Orthotropic Fail material model, [307](#)
- in Elastic Orthotropic material model, [303](#)

S AXIS

- in User Mount Section, [424](#)

S AXIS VARIABLE

- in User Mount Section, [424](#)

S DISPLACEMENT FUNCTION

- in Line Weld, [585](#)

S DISPLACEMENT SCALE FACTOR

- in Line Weld, [585](#)

S ROTATION FUNCTION

- in Line Weld, [585](#)

S ROTATION SCALE FACTOR

- in Line Weld, [585](#)

SCALAR SOURCE VARIABLE

- in Pressure, [533](#)
 - description of, [538](#)

SCALE FACTOR

- in Gravity, [558](#)
- in Initial Condition, [489](#)
 - description of, [495](#)
- in Lanczos Parameters, [116](#)
- in Loadstep Predictor, [188](#)
 - description of, [189](#)
- in Periodic, [512](#)
- in Pore Pressure, [566](#)
 - description of, [569](#)
- in Power Method Parameters, [123](#)
- in Prescribed Acceleration, [508](#)
 - description of, [511](#)
- in Prescribed Displacement, [499](#)
 - description of, [503](#)
- in Prescribed Force, [545](#)
 - description of, [549](#)
- in Prescribed Moment, [550](#)
 - description of, [553](#)
- in Prescribed Rotation, [516](#)
 - description of, [520](#)
- in Prescribed Rotational Velocity, [520](#)
 - description of, [524](#)
- in Prescribed Temperature, [561](#)
 - description of, [564](#)
- in Prescribed Velocity, [503](#)
 - description of, [507](#)
- in Pressure, [533](#)
 - description of, [539](#)
- in Reference Axis, [524](#)
- in REFERENCE AXIS ROTATION
 - usage in, [527](#)
- in Stiff Elastic material model, [335](#)
- in Traction, [541](#)
 - description of, [544](#)
- in Volume Repulsion Old, [594](#)

SCALING FUNCTION

- in Incompressible Solid material model, [329](#)

SEARCH CONFIGURATION

- in TIED MPC, [595](#)

SEARCH OPTIONS

- in Contact Definition
 - description of, [659](#)

SEARCH TOLERANCE

- in Contact Definition – in Search Options, [659](#)
 - description of, [659](#)
- in Line Weld, [585](#)
- in Periodic, [512](#)
- in Spot Weld, [579](#)
- in TIED MPC, [595](#)

SECOND ALPHA

- in Elastic 3D Orthotropic material model, [297](#)

- in Hill Plasticity material model, 252
- SECOND DIRECTION FOR ROTATION
 - in Elastic 3D Orthotropic material model, 297
 - in Hill Plasticity material model, 252
- SECOND SURFACE
 - in Spot Weld, 579
- SECTION
 - in Beam Section, 401
 - in Finite Element Model – in Parameters For Block, 363
 - description of, 366
 - general overview, 365
- Section command blocks
 - about, 375
- SELF CONTACT
 - in Contact Definition – in Interaction Defaults, 634
 - description of, 635
- SEMI IMPLICIT PLASTIC STRAIN SOLVER
 - NUMBER OF ITERATIONS
 - in BCJ_MEM material model, 271
- SEMI IMPLICIT PLASTIC STRAIN SOLVER
 - RESIDUAL TOLERANCE
 - in BCJ_MEM material model, 271
- SEND BLOCKS
 - in Procedural Transfer, 864
 - description of, 866
- SEND COORDINATES
 - in Procedural Transfer, 864
 - description of, 866
- SET RCB DIRECTIONS
 - in Zoltan Parameters, 470
- SHEAR
 - in Hyperfoam material model, 295
- SHEAR EXPONENT
 - in Foam Plasticity material model, 289
- SHEAR HARDENING
 - in Foam Plasticity material model, 289
- SHEAR MODULUS
 - in Barlat Plasticity material model, 258
 - in BCJ material model, 268
 - in Ductile Fracture material model, 231
 - in Elastic 3D Orthotropic material model, 297
 - in Elastic Fracture material model, 222
 - in Elastic material model, 217
 - in Elastic Orthotropic material model, 303
 - in Elastic Orthotropic Fail material model, 307
 - in Elastic-Plastic material model, 224
 - in Elastic-Plastic Power-Law Hardening material model, 228
 - in Foam Plasticity material model, 289
 - in Hill Plasticity material model, 252
 - in Hosford Plasticity material model, 248
 - in Hyperfoam material model, 295
 - in Incompressible Solid material model, 329
 - in Johnson–Cook material model, 265
 - in Low Density Foam material model, 293
 - in Multilinear Elastic-Plastic Hardening Model material model, 234
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, 238
 - in Neo Hookean material model, 220
 - in Orthotropic Crush material model, 317
 - in Orthotropic Rate material model, 321
 - in Plane Stress Rate Plasticity material model, 324
 - in Power Law Creep material model, 279
 - in Soil and Crushable Foam material model, 286
 - in Stiff Elastic material model, 335
 - in Thermoelastic material model, 219
 - in Thermoelastic Plastic material model, 244
 - in Viscoplastic material model, 281
 - in Wire Mesh material model, 315
- SHEAR MODULUS AB
 - in Elastic 3D Orthotropic material model, 297
- SHEAR MODULUS BC
 - in Elastic 3D Orthotropic material model, 297
- SHEAR MODULUS CA
 - in Elastic 3D Orthotropic material model, 297
- SHEAR RELAX TIME 1
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 10
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 2
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 3
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 4
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 5
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 6
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 7
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 8
 - in Viscoelastic Swanson material model, 340
- SHEAR RELAX TIME 9
 - in Viscoelastic Swanson material model, 340
- SHEAR STRENGTH
 - in Foam Plasticity material model, 289
- SHEAR_DAMAGE_MODULUS_12
 - in Elastic Orthotropic Fail material model, 307
- SHEAR_DAMAGE_MODULUS_13
 - in Elastic Orthotropic Fail material model, 307
- SHEAR_DAMAGE_MODULUS_23
 - in Elastic Orthotropic Fail material model, 307
- SHEAR_FIBER_STRENGTH_12
 - in Elastic Orthotropic Fail material model, 307
- SHEAR_FIBER_STRENGTH_13
 - in Elastic Orthotropic Fail material model, 307

- in Elastic Orthotropic Fail material model, [307](#)
- SHEAR_FIBER_STRENGTH_23
 - in Elastic Orthotropic Fail material model, [307](#)
- SHEAR_FRACTURE_ENERGY_12
 - in Elastic Orthotropic Fail material model, [307](#)
- SHEAR_FRACTURE_ENERGY_13
 - in Elastic Orthotropic Fail material model, [307](#)
- SHEAR_FRACTURE_ENERGY_23
 - in Elastic Orthotropic Fail material model, [307](#)
- SHEAR_MATRIX_STRENGTH_12
 - in Elastic Orthotropic Fail material model, [307](#)
- SHEAR_MATRIX_STRENGTH_13
 - in Elastic Orthotropic Fail material model, [307](#)
- SHEAR_MATRIX_STRENGTH_23
 - in Elastic Orthotropic Fail material model, [307](#)
- SHELL SECTION, [388](#)
- SIERRA, [65](#)
- SILENT BOUNDARY, [578](#)
- SKIN ALL BLOCKS
 - in Contact Definition
 - description of, [614](#)
- SLAVE
 - in Contact Definition – in Interaction, [637](#)
 - description of, [638](#)
 - in Periodic, [512](#)
- SLAVE BLOCK
 - in MPC, [598](#)
- SLAVE NODE SET
 - in MPC, [598](#)
- SLAVE NODES
 - in MPC, [598](#)
- SLAVE SURFACE
 - in MPC, [598](#)
- SLIP SCALE FACTOR
 - in Loadstep Predictor, [188](#)
 - description of, [189](#)
- SMALL NUMBER OF ITERATIONS
 - in Full Tangent Preconditioner, [153](#)
 - description of, [157](#)
- SMALL STRAIN
 - in Lanczos Parameters, [116](#)
 - in Power Method Parameters, [123](#)
- SOLID SECTION, [378](#)
- SOLVER, [135](#)
- SPECIFIC HEAT
 - in BCJ material model, [268](#)
- SPECIFIC HEAT FOR PLASTIC DISSIPATION CALCULATIONS
 - in BCJ_MEM material model, [271](#)
- SPH ALPHAQ PARAMETER
 - in Particle Section, [417](#)
- SPH BETAQ PARAMETER
 - in Particle Section, [417](#)
- SPH DECOUPLE STRAINS
 - in SPH Options, [419](#)
- SPH OPTIONS, [419](#)
- SPH SYMMETRY PLANE
 - in SPH Options, [419](#)
- SPHERE INITIAL RADIUS
 - in Particle Section, [417](#)
- SPOT WELD, [579](#)
- SPRING SECTION, [412](#)
- STAGNATION THRESHOLD
 - in Full Tangent Preconditioner, [153](#)
 - description of, [160](#)
- START TIME
 - in Heartbeat Output, [734](#)
 - in History Output, [728](#)
 - in Output Scheduler, [757](#)
 - in Restart Data, [745](#)
 - in Results Output, [696](#)
 - description of, [707](#)
 - in Time Stepping Block, [102](#), [191](#)
- STARTING VECTOR
 - in Lanczos Parameters, [116](#)
 - in Power Method Parameters, [123](#)
- State-Based Linear Peridynamic Solid Material Model
 - in Peridynamics, [788](#)
- STEP INTERVAL
 - in Node Based Time Step Parameters, [126](#)
 - in Parameters For Presto Region, [103](#)
 - description of, [104](#)
 - in Rebalance
 - description of, [469](#)
- STIFFNESS DAMPING COEFFICIENT
 - in Viscous Damping, [590](#)
 - description of, [591](#)
- STRAIN INCREMENTATION
 - in Solid Section, [378](#)
- STREAM NAME
 - in Heartbeat Output, [734](#)
- STRESS DECAY FUNCTION
 - in Peridynamics Section, [421](#)
- STRUCTURE NAME
 - in Mass Properties, [448](#)
 - description of, [449](#)
- Submodel, [600](#)
- SUBROUTINE DEBUGGING OFF
 - as user subroutine command line, [826](#)
 - description of, [827](#)
 - in Element Death, [450](#)
 - usage in, [457](#)
 - in Initial Condition, [489](#)
 - usage in, [494](#)
 - in Initial Velocity, [529](#)
 - usage in, [532](#)
 - in Pore Pressure, [566](#)
 - usage in, [568](#)

- in Prescribed Acceleration, 508
 - usage in, 511
- in Prescribed Displacement, 499
 - usage in, 502
- in Prescribed Force, 545
 - usage in, 548
- in Prescribed Moment, 550
 - usage in, 553
- in Prescribed Rotation, 516
 - usage in, 519
- in Prescribed Rotational Velocity, 520
 - usage in, 523
- in Prescribed Temperature, 561
 - usage in, 563
- in Prescribed Velocity, 503
 - usage in, 507
- in Pressure, 533
 - usage in, 536
- in Time Step Initialization, 828
 - usage in, 830
- in Traction, 541
 - usage in, 543
- in User Output, 711
 - usage in, 722

SUBROUTINE DEBUGGING ON

- as user subroutine command line, 826
- description of, 827
- in Element Death, 450
 - usage in, 457
- in Initial Condition, 489
 - usage in, 494
- in Initial Velocity, 529
 - usage in, 532
- in Pore Pressure, 566
 - usage in, 568
- in Prescribed Acceleration, 508
 - usage in, 511
- in Prescribed Displacement, 499
 - usage in, 502
- in Prescribed Force, 545
 - usage in, 548
- in Prescribed Moment, 550
 - usage in, 553
- in Prescribed Rotation, 516
 - usage in, 519
- in Prescribed Rotational Velocity, 520
 - usage in, 523
- in Prescribed Temperature, 561
 - usage in, 563
- in Prescribed Velocity, 503
 - usage in, 507
- in Pressure, 533
 - usage in, 536
- in Time Step Initialization, 828

- usage in, 830
- in Traction, 541
 - usage in, 543
- in User Output, 711
 - usage in, 722

SUBROUTINE INTEGER PARAMETER

- as user subroutine command line, 826
- description of, 827
- in Element Death, 450
 - usage in, 457
- in Initial Condition, 489
 - usage in, 494
- in Initial Velocity, 529
 - usage in, 532
- in Pore Pressure, 566
 - usage in, 568
- in Prescribed Acceleration, 508
 - usage in, 511
- in Prescribed Displacement, 499
 - usage in, 502
- in Prescribed Force, 545
 - usage in, 548
- in Prescribed Moment, 550
 - usage in, 553
- in Prescribed Rotation, 516
 - usage in, 519
- in Prescribed Rotational Velocity, 520
 - usage in, 523
- in Prescribed Temperature, 561
 - usage in, 563
- in Prescribed Velocity, 503
 - usage in, 507
- in Pressure, 533
 - usage in, 536
- in Time Step Initialization, 828
 - usage in, 830
- in Traction, 541
 - usage in, 543
- in User Output, 711
 - usage in, 722

SUBROUTINE REAL PARAMETER

- as user subroutine command line, 826
- description of, 827
- in Element Death, 450
 - usage in, 457
- in Initial Condition, 489
 - usage in, 494
- in Initial Velocity, 529
 - usage in, 532
- in Pore Pressure, 566
 - usage in, 568
- in Prescribed Acceleration, 508
 - usage in, 511
- in Prescribed Displacement, 499

- usage in, 502
- in Prescribed Force, 545
 - usage in, 548
- in Prescribed Moment, 550
 - usage in, 553
- in Prescribed Rotation, 516
 - usage in, 519
- in Prescribed Rotational Velocity, 520
 - usage in, 523
- in Prescribed Temperature, 561
 - usage in, 563
- in Prescribed Velocity, 503
 - usage in, 507
- in Pressure, 533
 - usage in, 536
- in Time Step Initialization, 828
 - usage in, 830
- in Traction, 541
 - usage in, 543
- in User Output, 711
 - usage in, 722
- usage with query function, 802

SUBROUTINE STRING PARAMETER

- as user subroutine command line, 826
- description of, 827
- in Element Death, 450
 - usage in, 457
- in Initial Condition, 489
 - usage in, 494
- in Initial Velocity, 529
 - usage in, 532
- in Pore Pressure, 566
 - usage in, 568
- in Prescribed Acceleration, 508
 - usage in, 511
- in Prescribed Displacement, 499
 - usage in, 502
- in Prescribed Force, 545
 - usage in, 548
- in Prescribed Moment, 550
 - usage in, 553
- in Prescribed Rotation, 516
 - usage in, 519
- in Prescribed Rotational Velocity, 520
 - usage in, 523
- in Prescribed Temperature, 561
 - usage in, 563
- in Prescribed Velocity, 503
 - usage in, 507
- in Pressure, 533
 - usage in, 536
- in Time Step Initialization, 828
 - usage in, 830
- in Traction, 541
 - usage in, 543
- in User Output, 711
 - usage in, 722

Subsetting

- in Results Output, 705

SUMMARY OUTPUT STEP INTERVAL

- in Element Death, 450
 - description of, 458

SUMMARY OUTPUT TIME INTERVAL

- in Element Death, 450
 - description of, 458

Support, 63

SURFACE

- in Fluid Pressure, 570
 - description of, 58, 482
- in Adaptive Refinement, 476
 - usage in, 477
- in Cavity Expansion , 576
- in Centripetal Force, 560
- in Contact Definition – in Contact Surface(block)
 - usage in, 612
- in Fixed Displacement, 497
 - usage in, 498
- in Fixed Rotation, 514
 - usage in, 515
- in Gravity, 558
 - usage in, 558
- in Inertia Relief, 573
 - usage in, 573
- in Initial Condition, 489
 - usage in, 490
- in Initial Velocity, 529
 - usage in, 530
- in Line Weld, 585
- in Mass Scaling, 128
 - usage in, 129
- in Prescribed Acceleration, 508
 - usage in, 509
- in Prescribed Displacement, 499
 - usage in, 500
- in Prescribed Force, 545
 - usage in, 547
- in Prescribed Moment, 550
 - usage in, 551
- in Prescribed Rotation, 516
 - usage in, 518
- in Prescribed Rotational Velocity, 520
 - usage in, 522
- in Prescribed Velocity, 503
 - usage in, 505
- in Pressure, 533
- in Silent Boundary, 578
- in Spot Weld, 579
- in Time Step Initialization, 828

- usage in, 829
- in Traction, 541
- in User Output, 711
 - usage in, 715
- in Volume Repulsion Old
 - usage in, 595
- in Volume Repulsion Old – in BLOCK SET, 594
- SURFACE EFFECT
 - in Cavity Expansion – in Layer, 576
 - description of, 578
- SURFACE SUBROUTINE
 - as user subroutine command line, 826
 - description of, 827
 - in Initial Condition, 489
 - description of, 494
 - in Pressure, 533
 - description of, 536
 - in Time Step Initialization, 828
 - description of, 830
 - in User Output, 711
 - description of, 721
- SURFACE VARIABLES
 - in Results Output, 696
- SURFACES
 - in Contact Definition – in Interaction, 637
 - description of, 638
 - in Contact Definition – in Interaction Defaults, 634
 - description of, 635
- SYNCHRONIZE OUTPUT
 - in Heartbeat Output, 734
 - description of, 739
 - in History Output, 728
 - description of, 732
 - in Restart Output, 745
 - description of, 755
 - in Results Output, 696
 - description of, 710
- SYSTEM
 - in Orientation, 80
- T AXIS
 - in Beam Section, 401
 - in User Mount Section, 424
- T AXIS VARIABLE
 - in User Mount Section, 424
- T DISPLACEMENT FUNCTION
 - in Line Weld, 585
- T DISPLACEMENT SCALE FACTOR
 - in Line Weld, 585
- T FUNCTION
 - in Orthotropic Rate material model, 321
- T ROTATION FUNCTION
 - in Line Weld, 585
- T ROTATION SCALE FACTOR
 - in Line Weld, 585
- TANGENT DIAGONAL SCALE
 - in Full Tangent Preconditioner, 153
 - description of, 155
- TANGENT DIAGONAL SHIFT
 - in Full Tangent Preconditioner, 153
 - description of, 155
- TANGENTIAL DISPLACEMENT FUNCTION
 - in Spot Weld, 579
- TANGENTIAL DISPLACEMENT SCALE FACTOR
 - in Spot Weld, 579
- TANGENTIAL TOLERANCE
 - in Contact Definition – in Interaction, 637
 - description of, 641
 - in Contact Definition – in Search Options, 659
 - description of, 659
- TARGET E
 - in Incompressible Solid material model, 329
 - in Mooney-Rivlin material model, 332
 - in Swanson material model, 337
 - in Viscoelastic Swanson material model, 340
- TARGET E FUNCTION
 - in Mooney-Rivlin material model, 332
 - in Swanson material model, 337
 - in Viscoelastic Swanson material model, 340
- TARGET ITERATIONS
 - in Adaptive Time Stepping
 - description of, 195
- TARGET PRESSURE INCREMENT
 - in Control Stiffness, 177
 - description of, 178
- TARGET RELATIVE CONTACT RESIDUAL
 - in Control Contact, 167
 - description of, 168
- TARGET RELATIVE PRESSURE INCREMENT
 - in Control Stiffness, 177
 - description of, 178
- TARGET RELATIVE RESIDUAL
 - in CG, 140
 - description of, 142
 - in Control Contact, 167
 - description of, 168
 - in Control Stiffness, 177
 - description of, 178
- TARGET RELATIVE SDEV INCREMENT
 - in Control Stiffness, 177
 - description of, 178
- TARGET RELATIVE STRAIN INCREMENT
 - in Control Stiffness, 177
 - description of, 178
- TARGET RELATIVE STRESS INCREMENT
 - in Control Stiffness, 177
 - description of, 178
- TARGET RESIDUAL

- in CG, [140](#)
 - description of, [142](#)
- in Control Contact, [167](#)
 - description of, [168](#)
- in Control Stiffness, [177](#)
 - description of, [178](#)
- TARGET SDEV INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- TARGET SMOOTHING RELATIVE RESIDUAL
 - in Full Tangent Preconditioner, [153](#)
- TARGET SMOOTHING RESIDUAL
 - in Full Tangent Preconditioner, [153](#)
- TARGET STRESS INCREMENT
 - in Control Stiffness, [177](#)
 - description of, [178](#)
- TARGET TIME STEP
 - in Mass Scaling, [128](#)
 - description of, [130](#)
- TEMPERATURE OPTION
 - in BCJ_MEM material model, [271](#)
- TEMPERATURE TYPE
 - in Prescribed Temperature
 - description of, [563](#)
- TEMPERATURES
 - in Thermoelastic Plastic material model, [244](#)
- TEMPO
 - in BCJ material model, [268](#)
- TENSILE_DAMAGE_MODULUS_11
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_DAMAGE_MODULUS_22
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_DAMAGE_MODULUS_33
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_FIBER_STRENGTH_11
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_FIBER_STRENGTH_22
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_FIBER_STRENGTH_33
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_FRACTURE_ENERGY_11
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_FRACTURE_ENERGY_22
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_FRACTURE_ENERGY_33
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_MATRIX_STRENGTH_11
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_MATRIX_STRENGTH_22
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSILE_MATRIX_STRENGTH_33
 - in Elastic Orthotropic Fail material model, [307](#)
- TENSION
 - in Wire Mesh material model, [315](#)

- TENSION RELEASE
 - in Contact Definition – in Interaction, [637](#)
 - description of, [642](#)
- TENSION RELEASE FUNCTION
 - in Contact Definition – in Interaction, [637](#)
 - description of, [642](#)
- TERMINATION TIME
 - in Heartbeat Output, [734](#)
 - in History Output, [728](#)
 - in Output Scheduler, [757](#)
 - in Restart Data, [745](#)
 - in Results Output, [696](#)
 - description of, [707](#)
 - in Time Control, [102](#), [191](#)
- THERMAL CONSTANT
 - in Power Law Creep material model, [279](#)
 - in Viscoplastic material model, [281](#)
- THERMAL EXPANSION FUNCTION
 - in Mooney-Rivlin material model, [332](#)
 - in Swanson material model, [337](#)
 - in Viscoelastic Swanson material model, [340](#)
- THERMAL STRAIN AA FUNCTION
 - in Elastic 3D Orthotropic material model, [297](#)
- THERMAL STRAIN BB FUNCTION
 - in Elastic 3D Orthotropic material model, [297](#)
- THERMAL STRAIN CC FUNCTION
 - in Elastic 3D Orthotropic material model, [297](#)
- THERMAL_STRAIN_11_FUNCTION
 - in Elastic Orthotropic Fail material model, [307](#)
 - in Elastic Orthotropic material model, [303](#)
- THERMAL_STRAIN_22_FUNCTION
 - in Elastic Orthotropic Fail material model, [307](#)
 - in Elastic Orthotropic material model, [303](#)
- THERMAL_STRAIN_33_FUNCTION
 - in Elastic Orthotropic Fail material model, [307](#)
 - in Elastic Orthotropic material model, [303](#)
- THETA
 - in Periodic, [512](#)
- THETA OPT
 - in BCJ material model, [268](#)
- THICKNESS
 - in Membrane Section, [398](#)
 - in Shell Section, [388](#)
- THICKNESS MESH VARIABLE
 - in Membrane Section, [398](#)
 - in Shell Section, [388](#)
- THICKNESS SCALE FACTOR
 - in Membrane Section, [398](#)
 - in Shell Section, [388](#)
- THICKNESS TIME STEP
 - in Membrane Section, [398](#)
 - in Shell Section, [388](#)
- TIED FACES
 - in TIED MPC, [595](#)

- TIED MODEL
 - in Contact Definition
 - description of, 625
- TIED MPC, 595
- TIED NODES
 - in TIED MPC, 595
- TIME
 - in Initial Condition, 489
 - in Pore Pressure, 566
 - description of, 568
 - in Prescribed Acceleration, 508
 - description of, 511
 - in Prescribed Displacement, 499
 - description of, 503
 - in Prescribed Rotation, 516
 - description of, 519
 - in Prescribed Rotational Velocity, 520
 - description of, 524
 - in Prescribed Temperature, 561
 - description of, 563
 - in Prescribed Velocity, 503
 - description of, 507
 - in Pressure, 533
 - description of, 537
- TIME CONTROL
 - example of, 105, 198
 - in Adagio Procedure, 87
 - contents and description of, 191
 - overview, 191
 - in Presto Procedure
 - general layout of, 100
 - in Presto Procedure, 87
 - contents and description of, 102
 - overview, 99
 - usage of, 100
 - in Procedure
 - about, 88
- TIME INCREMENT
 - in Parameters For Adagio Region, 192, 193
- TIME INCREMENT FUNCTION
 - in Parameters For Adagio Region, 192, 193
- TIME STEP INCREASE FACTOR
 - in Parameters For Presto Region, 103
 - in Parameters For Presto Region
 - description of, 104
 - usage of, 101
- TIME STEP INITIALIZATION, 828
- TIME STEP LIMIT
 - in Node Based Time Step Parameters, 126
- TIME STEP SCALE FACTOR
 - in Parameters For Presto Region, 103
 - in Parameters For Presto Region
 - description of, 104
 - usage of, 101
 - with Element Numerical Formulation, 371
 - with Peridynamics, 793
- TIME STEPPING BLOCK
 - in Time Control, 102
- TIME STEPPING BLOCK
 - in Time Control, 191
 - description of, 102, 191
- TIMESTEP ADJUSTMENT INTERVAL
 - in Heartbeat Output, 734
 - in History Output, 728
 - in Output Scheduler, 757
 - in Restart Data, 745
 - in Results Output, 696
 - description of, 707
- TIMESTEP FORMAT
 - in Heartbeat Output, 734
- TIP RADIUS
 - in Cavity Expansion, 576
- TITLE, 65
 - in Heartbeat Output, 734
 - in History Output, 728
 - in Results Output, 696
- TL FUNCTION
 - in Orthotropic Rate material model, 321
- TOLERANCE
 - in Plane Stress Rate Plasticity material model, 324
- TORQUE
 - in Reference Axis, 524
 - in Reference Axis Rotation
 - description of, 527
- TRACTION, 541
- TRACTION DISPLACEMENT FUNCTION
 - in Contact Definition – in Cohesive Zone Model
 - description of, 626
- TRACTION DISPLACEMENT SCALE FACTOR
 - in Contact Definition – in Cohesive Zone Model
 - description of, 626
- TRACTION INTEGRATION METHOD
 - in Traction, 541
- Transfers, 863
- TRANSFORMATION TYPE
 - in Procedural Transfer, 864
 - description of, 866
- TRANSVERSE SHEAR HOURGLASS STIFFNESS
 - in Finite Element Model – in Parameters For Block, 363
 - description of, 367
- TRANSVERSE SHEAR HOURGLASS VISCOSITY
 - in Finite Element Model – in Parameters For Block, 363
 - description of, 367
- TRUSS SECTION, 411
- TX
 - in Orthotropic Rate material model, 321

TY
 in Orthotropic Rate material model, 321

TYPE
 in Coordinate System, 84
 in Function, 67
 in Loadstep Predictor, 188
 in Time Step Initialization
 description of, 830
 in User Variable, 831

TZ
 in Orthotropic Rate material model, 321

UPDATE ON TIME STEP CHANGE
 in Lanczos Parameters, 116
 in Power Method Parameters, 123

UPDATE STEP INTERVAL
 in Lanczos Parameters, 116
 in Power Method Parameters, 123

USE FINITE ELEMENT MODEL, 90

USE HHT INTEGRATION
 in Implicit Dynamics, 199
 description of, 199

USE OUTPUT SCHEDULER
 example of, 758
 in Heartbeat Output, 734
 description of, 739
 in History Output, 728
 description of, 733
 in Restart Data, 745
 description of, 755
 in Results Output, 696
 description of, 710

USE WITH RESTART
 in User Variable, 831

USER CRITERIA GLOBAL
 in Adaptive Time Stepping
 description of, 198

USER INTEGRATION RULE
 in Shell Section, 388

USER MOUNT SECTION, 424

USER OUTPUT, 711
 example of, 839

USER RATE DEPENDENCE
 in Plane Stress Rate Plasticity material model, 324

USER SUBROUTINE FILE, 67
 example of, 834
 usage in context, 826

User Subroutines
 aupst_check_elem_var, 806
 aupst_check_global_var, 816
 aupst_check_node_var, 806
 aupst_cyl_transform, 855
 aupst_evaluate_function, 805
 aupst_get_elem_centroid, 823
 aupst_get_elem_nodes, 818
 aupst_get_elem_topology, 818
 aupst_get_elem_var, 806
 aupst_get_elem_var_offset, 806
 aupst_get_face_nodes, 818
 aupst_get_face_topology, 818
 aupst_get_global_var, 816
 aupst_get_integer_param, 800
 aupst_get_node_var, 806
 aupst_get_point, 823
 aupst_get_proc_num, 823
 aupst_get_real_param, 800
 aupst_get_string_param, 800
 aupst_get_time, 805
 aupst_put_elem_var, 806
 aupst_put_elem_var_offset, 806
 aupst_put_global_var, 816
 aupst_put_node_var, 806
 aupst_rec_transform, 857
 copy_data, 858
 ELEMENT BLOCK SUBROUTINE, 826
 HEARTBEAT OUTPUT, 734
 HISTORY OUTPUT, 728
 NODE SET SUBROUTINE, 826
 RESULTS OUTPUT, 696
 SUBROUTINE DEBUGGING OFF, 826
 SUBROUTINE DEBUGGING ON, 826
 SUBROUTINE INTEGER PARAMETER, 826
 SUBROUTINE REAL PARAMETER, 826
 SUBROUTINE STRING PARAMETER, 826
 SURFACE SUBROUTINE, 826
 USER OUTPUT, 711
 USER VARIABLE, 831

USER VARIABLE, 831

VALUES
 in Function, 67

VARIABLE TYPE
 in Initial Condition, 489
 description of, 491

VECTOR
 in Coordinate System, 84

VECTOR NODE
 in Coordinate System, 84

VECTOR SCALE
 in Lanczos Parameters, 116
 in Power Method Parameters, 123

VELOCITY DAMPING COEFFICIENT
 in Pressure, 533
 description of, 535
 in Viscous Damping
 description of, 591

VISCOUS DAMPING, 590

VMIN

- in Orthotropic Crush material model, [317](#)
- Void Elements, [379](#)
- VOLUME
 - in Point Mass Section, [414](#)
- VOLUME REPULSION OLD, [594](#)
- VOLUMETRIC EFFECTS OF DAMAGE OPTION
 - in BCJ_MEM material model, [271](#)
- VOLUMETRIC SEARCH TOLERANCE
 - in TIED MPC, [595](#)
- W FUNCTION
 - in Orthotropic Rate material model, [321](#)
- WALL THICKNESS
 - in Beam Section, [401](#)
- WEIBULL
 - in Initial Condition, [489](#)
 - description of, [492](#)
- WEIBULL MEDIAN
 - in Initial Condition, [489](#)
- WEIBULL SCALE
 - in Initial Condition, [489](#)
- WEIBULL SCALING EXPONENT SCALE
 - in Initial Condition, [489](#)
- WEIBULL SCALING FIELD NAME
 - in Initial Condition, [489](#)
- WEIBULL SCALING FIELD TYPE
 - in Initial Condition, [489](#)
- WEIBULL SCALING REFERENCE VALUE
 - in Initial Condition, [489](#)
- WEIBULL SEED
 - in Initial Condition, [489](#)
- WEIBULL SHAPE
 - in Initial Condition, [489](#)
- WEIGHTS
 - in RBE3, [601](#)
- WIDTH
 - in Beam Section, [401](#)
- WLF COEF C1
 - in Viscoelastic Swanson material model, [340](#)
- WLF COEF C2
 - in Viscoelastic Swanson material model, [340](#)
- WLF TREF
 - in Viscoelastic Swanson material model, [340](#)
- WT FUNCTION
 - in Orthotropic Rate material model, [321](#)
- YIELD CURVES
 - in Plane Stress Rate Plasticity material model, [324](#)
- YIELD FUNCTION
 - in Wire Mesh material model, [315](#)
- YIELD STRAIN RATES
 - in Plane Stress Rate Plasticity material model, [324](#)
- YIELD STRESS
 - in Ductile Fracture material model, [231](#)
 - in Elastic-Plastic material model, [224](#)
 - in Elastic-Plastic Power-Law Hardening material model, [228](#)
 - in Johnson–Cook material model, [265](#)
 - in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
 - in Orthotropic Crush material model, [317](#)
 - in Orthotropic Rate material model, [321](#)
 - in Plane Stress Rate Plasticity material model, [324](#)
 - in Thermoelastic Plastic material model, [244](#)
- YIELD STRESS FUNCTION
 - in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
 - in Thermoelastic Plastic material model, [244](#)
- YOUNGS MODULUS
 - in Barlat Plasticity material model, [258](#)
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
 - in Ductile Fracture material model, [231](#)
 - in Elastic 3D Orthotropic material model, [297](#)
 - in Elastic Fracture material model, [222](#)
 - in Elastic material model, [217](#)
 - in Elastic Orthotropic Fail material model, [307](#)
 - in Elastic Orthotropic material model, [303](#)
 - in Elastic-Plastic material model, [224](#)
 - in Elastic-Plastic Power-Law Hardening material model, [228](#)
 - in Foam Plasticity material model, [289](#)
 - in Hill Plasticity material model, [252](#)
 - in Hosford Plasticity material model, [248](#)
 - in Hyperfoam material model, [295](#)
 - in Incompressible Solid material model, [329](#)
 - in Johnson–Cook material model, [265](#)
 - in Low Density Foam material model, [293](#)
 - in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
 - in Neo Hookean material model, [220](#)
 - in Orthotropic Crush material model, [317](#)
 - in Orthotropic Rate material model, [321](#)
 - in Plane Stress Rate Plasticity material model, [324](#)
 - in Power Law Creep material model, [279](#)
 - in Soil and Crushable Foam material model, [286](#)
 - in Stiff Elastic material model, [335](#)
 - in Swanson material model, [337](#)
 - in Thermoelastic material model, [219](#)
 - in Thermoelastic Plastic material model, [244](#)
 - in Viscoelastic Swanson material model, [340](#)
 - in Viscoplastic material model, [281](#)

- in Wire Mesh material model, [315](#)
- YOUNGS MODULUS AA
 - in Elastic 3D Orthotropic material model, [297](#)
- YOUNGS MODULUS BB
 - in Elastic 3D Orthotropic material model, [297](#)
- YOUNGS MODULUS CC
 - in Elastic 3D Orthotropic material model, [297](#)
- YOUNGS MODULUS FUNCTION
 - in BCJ material model, [268](#)
 - in BCJ_MEM material model, [271](#)
 - in Multilinear Elastic-Plastic Hardening Model material model, [234](#)
 - in Multilinear Elastic-Plastic Hardening Model with Failure material model, [238](#)
 - in Thermoelastic material model, [219](#)
 - in Thermoelastic Plastic material model, [244](#)
- ZERO LENGTH R AXIS
 - in User Mount Section, [424](#)
- ZOLTAN DEBUG LEVEL
 - in Zoltan Parameters, [470](#)
 - as default, [470](#)
- ZOLTAN PARAMETERS
 - as command block, [470](#)
 - with defaults, [470](#)
 - as command line in Rebalance, [468](#)
 - description of, [470](#)

Distribution

1 0899 Technical Library, 9536 (1 electronic)

